Model Objects

AERMOD/ISC models sources, receptors, and buildings. Each of these model components are refereed to as **model objects** within BREEZE. The number of model objects that can be modeled is limited only to the amount of physical disk space and memory on your computer.

The most common method of creating model objects are:

- Using drawing tools in the Map View
- Enter data into the Spreadsheet / table view

Model object forms have "VCR" buttons that can be used to navigate among the model objects. They are:

Go to the First model object of this type

Go to the Previous model object of this type

Go to the Next model object of this type

Go to the Last model object of this type

Search for a specific model object ID

Add a new model object of this type

Remove or delete the current model object

Source Types

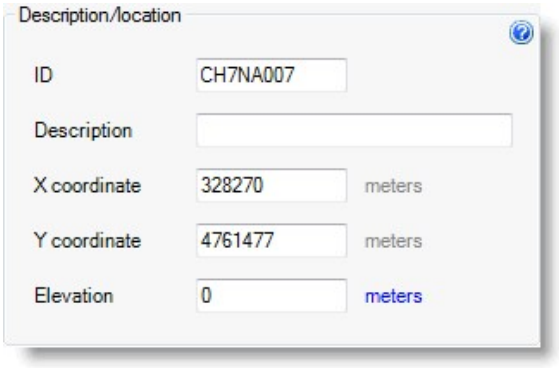
The following source types are available:

The Source screen allows you to choose the source type (point, flare, area, circular area, polygon area, volume, open pit, or line) and edit source parameters, building downwash data (point and flare sources only), emissions factors, particle data and pollutant emission rates (for the multi-pollutant option). The Source screen is automatically opened whenever you create a new source using Source button in the Model Objects group under the Project tab

, selecting Sources from the Objects menu under other tabs, or selecting the specific source type from the Sources menu in the Drawing group under the Map tab. The source types available are point, flare, area, circular area, polygon area, volume, open pit, and line.

For information on entering or editing building downwash, emission factors, particle data, or pollutant information, see the sections "Entering Building Downwash Information", "Entering Emission Factors Information", "Entering Particle Data Information" and "Entering Pollutants Information", respectively.

Description/location



ID: Enter a unique alphanumeric identifier for each source, up to eight characters. Each source ID should contain at least one alphabetical character. BREEZE will automatically create a unique ID for you when the source is initially created. Do not use a hyphen (-) or space when specifying a source ID (e.g., do not enter SOURCE-1). If you wish to use a separator, type an underscore "_" (e.g., SOURCE_1). Trailing spaces are ignored by the program.

NOTE: If you add a group of sources using Create Object Array tool, the Description/Location group box will not be editable at the time. Source IDs will be generated by BREEZE AERMOD at the beginning. You can change the source IDs later on from the Data Explorer view.

Description: Enter an optional description with additional source information, up to thirty characters.

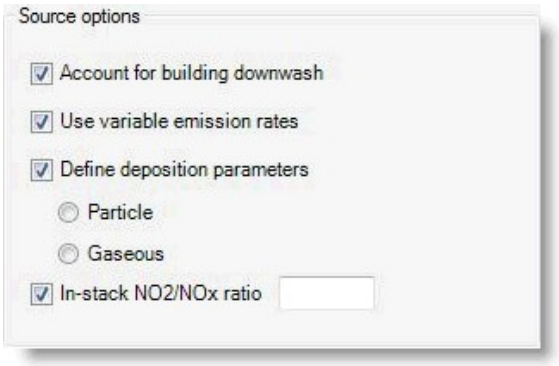
X and Y coordinate: Enter the X and Y coordinate of the source in meters.

For Point, Capped Point, Horizontal Point, Flare, Volume and Circular Area sources, X and Y are coordinates of the center of the source. This field is automatically populated with coordinates based on the mouse position when the mouse is clicked.

For Rectangular Area, Polygon Area, and Open Pit sources, X and Y are the coordinates of the first corner of the source as it is created in the Map view. It is typically referred to as the southwest corner of the source.

Elevation: Enter the base height of the source above mean sea level. This will be updated if sources are included during the importation of digital elevation model (DEM) data.

Source options



Account for building downwash: If checked, building downwash parameters have been specified for this source. These parameters can be viewed in the Building Downwash tab of the source form. The parameters are typically created by the BPIP program and not manually entered by the user.

Use variable emission rates: If checked, variable emission rates can be specified for this source. The variable emission rates tab is displayed when selected.

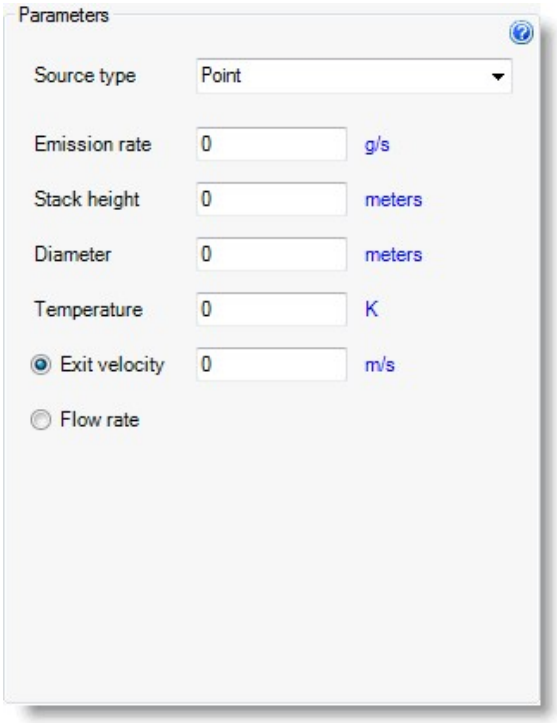
Define particle deposition parameters: If checked, particle deposition parameters can be defined for this source. For the 09292 executable, the user must select whether Particle or Gaseous deposition parameters will be entered. This selection will bring up the Particle Deposition or Gas Deposition tab in the source form.

In-stack NO2/NOx ratio: Present with the Pollutant ID is NO2. If checked, a source-specific ratio between 0 and 1 can be specified. This will override the default value set in the Ozone tab of the Control Options form.

Point

Point sources involve the release of emissions from a well-defined stack or vent, at known physical stack parameters and operational conditions. Consequently, characterizing point sources for modeling is fairly straightforward. The basic model inputs for any point source are listed below.

Parameters



Emission rate: Enter the pollutant emission rate of the source. This field is enabled only when multiple pollutants are not specified in the Pollutants tab of the Control Options form.

Stack height: Enter the emission release height above ground.

Diameter: Enter the inside diameter of the stack or vent.

Temperature: Enter the exit temperature of the stack gas.

- If a value of 0.0 Kelvin is input for the exit temperature, AERMOD will adjust the exit temperature for each hour to match the ambient temperature. This option allows the user to model a plume that is released at ambient temperature.
- The user may also model a plume with an exit temperature that exceeds the ambient temperature by a fixed amount by entering a negative value for exit temperature equal in magnitude to the temperature difference. The model will add the absolute value of a negative exit temperature to the ambient temperature for each hour to obtain the exit temperature used in computing the buoyancy flux of the plume.
- The AERMOD model does not include algorithms to model plumes that are released at temperatures below ambient temperature. Such releases should be modeled with a dense gas model. Dense gas models such as SLAB and DEGADIS can be found at www.breeze-software.com.

Exit velocity: Enter the stack gas exit velocity. You can choose to enter the flow rate if you do not want to enter this parameter.

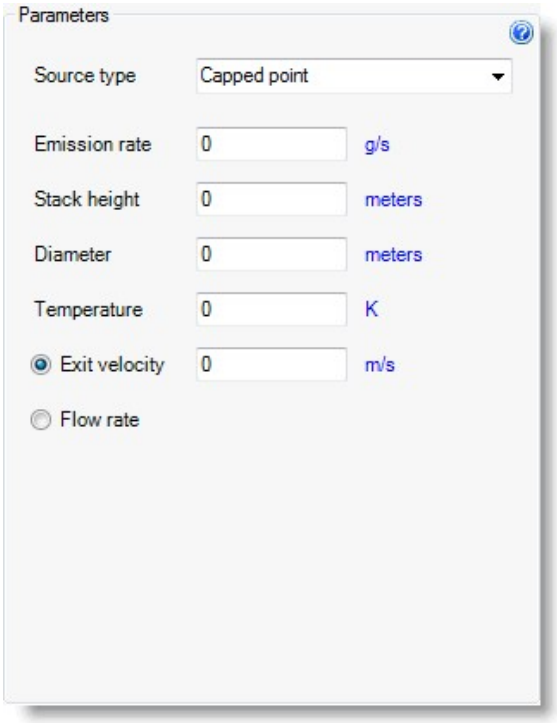
Flow rate: Enter the stack gas flow rate. You can choose to enter the exit velocity if you do not want to enter this parameter.

Capped point

The Capped Point source is currently a beta source type. For more on beta options, see the Enable Beta Options option in the Control Options form.

For capped point sources, specify the actual stack parameters [release height (m), exit temperature (K), exit velocity (m/s), and stack diameter (m)] as if the release were a non-capped vertical point source. The AERMOD model performs the necessary adjustments internally to account for plume rise and stack-tip downwash.

Parameters



Emission rate: Enter the pollutant emission rate of the source. This field is enabled only when multiple pollutants are not specified in the Pollutants tab of the Control Options form.

Stack height: Enter the emission release height above ground.

Diameter: Enter the inside diameter of the stack or vent.

Temperature: Enter the exit temperature of the stack gas.

- If a value of 0.0 Kelvin is input for the exit temperature, AERMOD will adjust the exit temperature for each hour to match the ambient temperature. This option allows the user to model a plume that is released at ambient temperature.
- The user may also model a plume with an exit temperature that exceeds the ambient temperature by a fixed amount by entering a negative value for exit temperature equal in magnitude to the temperature difference. The model will add the absolute value of a negative exit temperature to the ambient temperature for each hour to obtain the exit temperature used in computing the buoyancy flux of the plume.
- The AERMOD model does not include algorithms to model plumes that are released at temperatures below ambient temperature. Such releases should be modeled with a dense gas model. Dense gas models such as SLAB and DEGADIS can be found at www.breeze-software.com.

Exit velocity: Enter the stack gas exit velocity. You can choose to enter the flow rate if you do not want to enter this parameter.

Flow rate: Enter the stack gas flow rate. You can choose to enter the exit velocity if you do not want to enter this parameter.

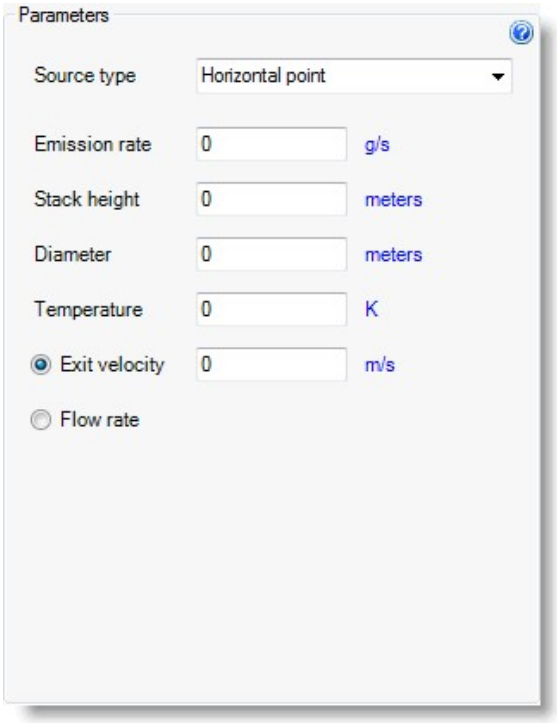
Horizontal point

The Horizontal Point source is currently a beta source type. For more on beta options, see the Enable Beta Options option in the Control Options form.

For horizontal point sources, specify the actual stack parameters [release height (m), exit temperature (K), exit velocity (m/s), and stack diameter (m)] as if the release were a non-capped vertical point source. The AERMOD model performs the necessary adjustments internally to account for plume rise. No stack-tip downwash is considered for horizontal release.

For horizontal releases, the model currently assumes that the release is oriented with the wind direction. For PRIME-downwashed sources, the user-specified exit velocity for horizontal releases is treated initially as horizontal momentum in the downwind direction.

Parameters



Emission rate: Enter the pollutant emission rate of the source. This field is enabled only when multiple pollutants are not specified in the Pollutants tab of the Control Options form.

Stack height: Enter the emission release height above ground.

Diameter: Enter the inside diameter of the stack or vent.

Temperature: Enter the exit temperature of the stack gas.

- If a value of 0.0 Kelvin is input for the exit temperature, AERMOD will adjust the exit temperature for each hour to match the ambient temperature. This option allows the user to model a plume that is released at ambient temperature.
- The user may also model a plume with an exit temperature that exceeds the ambient temperature by a fixed amount by entering a negative value for exit temperature equal in magnitude to the temperature difference. The model will add the absolute value of a negative exit temperature to the ambient temperature for each hour to obtain the exit temperature used in computing the buoyancy flux of the plume.
- The AERMOD model does not include algorithms to model plumes that are released at temperatures below ambient temperature. Such releases should be modeled with a dense gas model. Dense gas models such as SLAB and DEGADIS can be found at www.breeze-software.com.

Exit velocity: Enter the stack gas exit velocity. You can choose to enter the flow rate if you do not want to enter this parameter.

Flow rate: Enter the stack gas flow rate. You can choose to enter the exit velocity if you do not want to enter this parameter.

BREEZE Flare

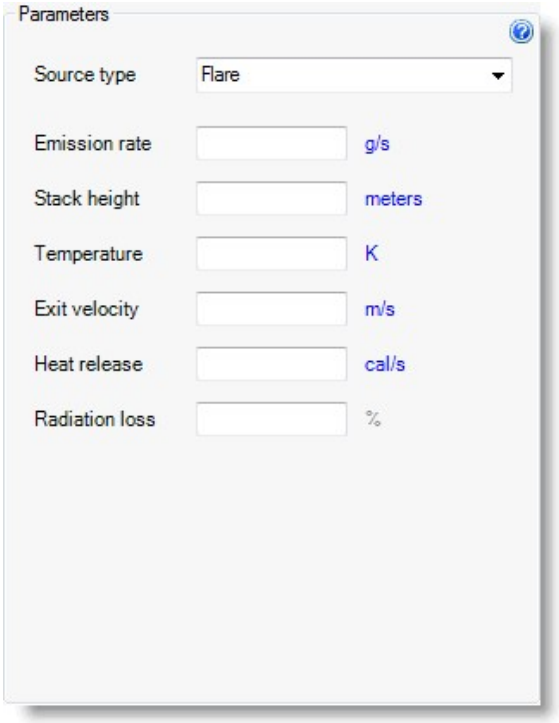
The flare source is not an EPA model available source type and is only available in the BREEZE version when the Regulatory Default option is not set in the Control Options form.

The BREEZE Enhanced Fortran executable must be selected for this source type to be used.

BREEZE ISC and AERMOD have the ability to model flares. Flares are open flames often located atop tall stacks that are used to dispose of waste processing gases from upsets or to regularly dispose of minor amounts of hydrogen sulfide at natural gas processing plants.

By knowing the composition of the flared gas, the volume of each constituent, and by using the low heat value for each constituent, the buoyancy flux of the total release can be determined. After subtracting energy lost to radiation, an effective height can be determined. Also, from the composition and volume of the flare gases and consideration of chemical transformations, such as, conversion of hydrogen sulfide to sulfur dioxide, the emission rate of pollutants can be calculated.

BREEZE ISC and AERMOD model point sources that require input of stack diameter, exit velocity, and stack gas temperature. Flare sources in BREEZE are modeled as pseudo point sources. No release diameter is required for a Flare source. BREEZE will determine the appropriate buoyancy flux based on the heat release for each hour modeled. The effective diameter of the source will change for each hour because it is a function of the difference in ambient and release temperature. The calculated effective diameter can be seen in the output *.lst file if the "Disp params" (dispersion parameters debug) option is check in the Output folder of the Control Options form.



Emission Rate: Enter the pollutant emission rate of the source. This field is enabled only when multiple pollutants are not specified in the Pollutants tab of Control Options form.

Stack Height: Enter the emission release height above ground.

Temperature: Enter the exit temperature of the stack gas. If this temperature is below ambient air temperature (as determined by the sequential meteorological data) the source temperature will default to the ambient air temperature for that hour. If the temperature is entered as a negative value in Kelvin, the model will default the release temperature to ambient air temperature plus (+) the absolute value of the release temperature. For example, if the release temperature is –10 Kelvin and the ambient air temperature is 283 Kelvin, the release temperature will default to 293 Kelvin.

Exit Velocity/Flow Rate: Select either the Exit velocity or Flow rate option and enter the stack gas exit velocity or flow rate, respectively.

Heat Release: Enter the gross heat released (calories/s or Btu/hr).

Radiation Loss: Enter the percentage of heat lost due to radiation.

Losses Due to Radiation

Most of the energy of combustion in a flare source is used to heat the products of combustion. A small portion is radiated. The visible and ultraviolet emissions rarely exceed 0.4 percent of the energy of combustion. The infrared emissions are much larger and often range from 10 to 15 percent in well-designed flares. The energy lost through infrared radiation is a function of the diameter of the burner and the luminescence for the flame. The typical fraction of total energy output that is radiated is 5 to 8 percent for hydrogen flames, 12 to 14 percent for hydrocarbon flames, and up to 18 percent for smoky flames burning larger molecules of hydrocarbons (Swander, 1988).

Judgment should be used to determine the amount of radiated energy. In order not to underestimate radiation and therefore overstate plume rise, a 20 percent loss is suggested. This means that 80 percent of the heat of combustion is used to heat the products of combustion. Leahey (1988) suggests that radiation losses can range from 20 to 55 percent of total energy but these observations were based on a flare made purposely smoky. Flares that contain unburned carbon will have much higher radiation losses than well-designed flares.

BREEZE SCREEN3 will make calculations for flares. This model assumes that 55 percent of the energy in the flare is lost by radiation, therefore, the plume rise estimated will be an underestimate for any source with less energy loss due to radiation. This will result in conservative estimates (overestimates) of ground-level concentrations.

How are Flares Handled in BREEZE

The flare parameters entered for a Flare source are used to determine the buoyancy flux (Fb). They are similar to a typical point source and are described below.

Emission Rate: The emission rate of resulting pollutants can be determined by converting the volumes burned to mass using the universal gas law and accounting for any chemical transformation. For example, if the volumetric flow rate of H2S is 0.0689 m3/sec, use the gas law to convert it to mass:

$$m = \frac{PVM}{RT} = \frac{(1013.5)(0.0689)(34)}{(0.0831)(293)} = 97.5\frac{g}{s}$$

The molecular weight of H2S is 34 and that of SO2 is 64. Therefore, (64/34)(97.5) = 185 g/s SO2 is produced by burning 97.5 of H2S.

Stack height: The stack height refers to the height above ground level where the physical stack ends.

Temperature: The average temperature of the flare.

Exit velocity: The rate at which the effluent exists the stack.

Heat release: The total heat released during combustion of the flare. Using values of the low heat value from appropriate engineering tables (Table 25, de Lorenzi 1952, Table 32, Marks, 1951), one can calculate the heat release for each constituent. This is shown in the table below:

Fuel	MMCF/Hour Burned	1. Low Heat Value BTU/CF at 68F 14.7 PSIA	Total Heat Release BTU/Hour
CH_4	0.00037	896	331520
C_2H_6	0.00010	1594	159400
C_3H_8	0.00008	2282	182560
C_4+	0.00013	≅ 3000	390000
CO	-	316	
H_2S	0.00826	560	4625600
Total	0.00894		5689080

Radiation loss: BREEZE will use the net heat released to determine Fb. This parameter is determined internally by supplying the percent of the total heat release that is lost by radiation. Enter a value between 0 and 100.

References:

de Lorenzi, O., 1952: Combustion Engineering, Superheater, Inc.

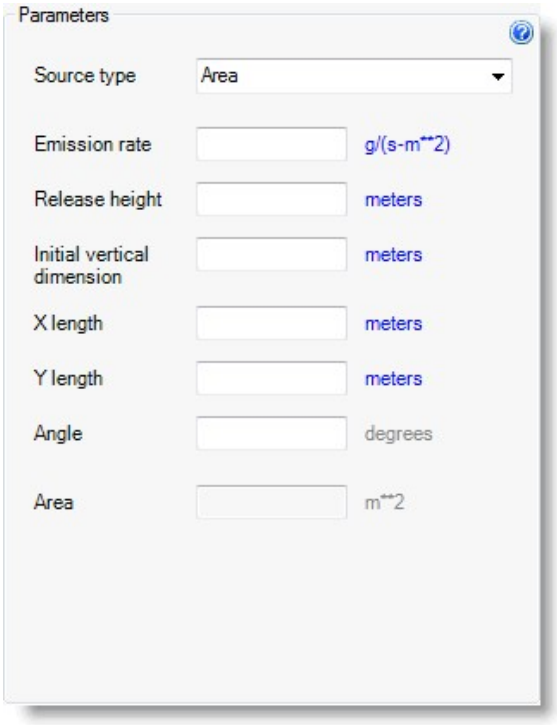
Leahey, D. M., 1988: Personal Communication – Letter to Richard Schulze (30 Aug. 88), Manager, Air Quality Services, Western Re3search, Calgary, Alberta, Canada.

Marks, L. S., 1951: Marks Mechanical Engineers Handbook, Maple Press, York, PA.

Swander, W., 1988: Personal communication – Richard Schulze with Walter Swander (7 Mar. 88) Manager, Flares/Resource Recovery, John Zink Co., Tulsa, OK 74170

Rectangular Area

Parameters



Emission rate: Enter the emission rate of the area source. The emission rate is per unit area; therefore, by adjusting the size of the area source, the emission rate of the source is adjusted as well.

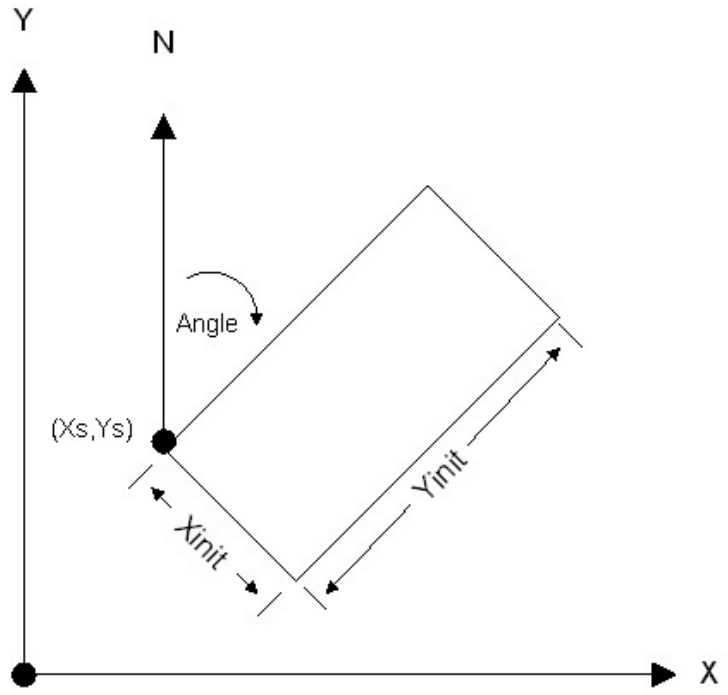
Release height: Enter the release height above ground level.

Initial vertical dimension: Enter the initial vertical dimension of the area source plume.

X length: Enter the length of the X side of the area source (in the east-west direction if the rotation angle is 0 degrees).

Y length: Enter the length of the Y side of the area source (in the north-south direction if the rotation angle is 0 degrees). If blank, Y length is assumed equal to X length; i.e., the area source is a square.

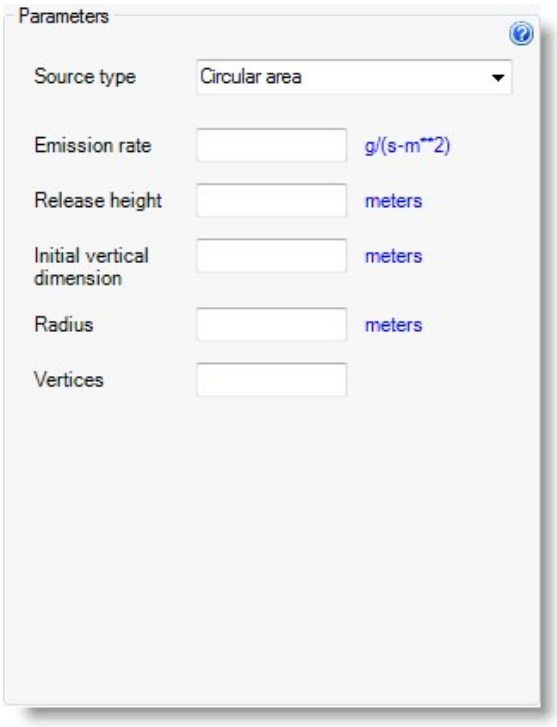
Angle: (Optional) Enter the rotation or orientation angle for the area source in degrees from North, measured positive in the clockwise direction. If left blank, it is assumed that the area source is oriented in the north-south and east-west directions (i.e., the rotation angle is 0 degrees). If a non-zero value is entered in this field, the model will rotate the area clockwise about the vertex. The vertex is defined by the X and Y coordinates that anchor the area source. The rotation angle may be positive (for clockwise rotation) or negative (for counterclockwise rotation), but the absolute value of the rotation angle cannot exceed 180 degrees.



Circular Area

The circular area source type may be used to specify an area source as a circular shape. The model will automatically generate a regular polygon to approximate the circular area source. The polygon will have the same area as that specified for the circle.

Parameters



Emission rate: Enter the emission rate of the circular area source. The emission rate is per unit area; therefore, by adjusting the size of the circular area source, the emission rate of the source is adjusted as well.

Release height: Enter the release height above ground.

Initial vertical dimension: (Optional) Enter the initial vertical dimension of the area source plume.

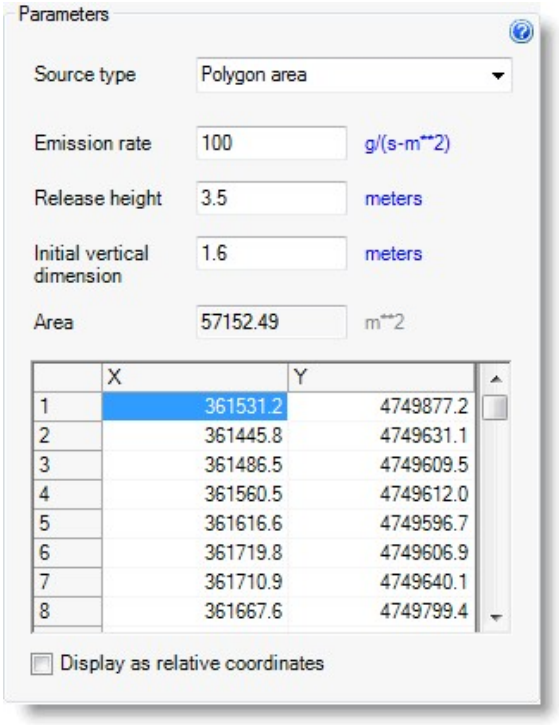
Radius: Enter the length of the circular area source from the center of the source to the outer edge of the source.

Vertices: Enter the number of vertices that make up the circular area source. A value must be larger than or equal to 3. By entering in a value of 3, the "circle" will be represented by an equilateral triangle.

Polygon Area

The polygon area source type may be used to specify an area source as an arbitrarily shaped polygon. This source type option provides the user with considerable flexibility for specifying the shape of an area source.

Parameters



Emission rate: Enter the emission rate of the polygon area source. The emission rate is per unit area; therefore, by adjusting the size of the polygon area source, the emission rate of the source is adjusted as well.

Release height: Enter the release height above ground.

Initial vertical dimension: (Optional) Enter the initial vertical dimension of the area source plume.

X and Y: Enter the X and Y distances from the X and Y coordinates (the sources handle) of the source location for each vertex of the polygon area source. These are not new coordinates, simply distances (either plus or minus). The first vertex must always be X = 0, Y = 0.

Display as relative coordinates: If checked, the X and Y coordinates are displayed as delta X and Y distances from the X and Y coordinate specified in the Location group box.

Volume

The volume source type are used to model releases from a variety of industrial sources, such as building roof monitors, multiple vents, and conveyor belts.

There are two basic types of volume sources: surface-based or ground-level sources that may also be modeled as area sources, and elevated sources. As with area sources, emissions from volume sources are assumed to be of neutral buoyancy. The effective

emission height of a surface-based volume source, such as a surface rail line, is usually set equal to zero. An example of an elevated volume source is an elevated conveyor with an effective emission height set equal to the height of the conveyor. A source may be defined as a volume source for modeling when its emissions can be considered to occur over a certain area and within a certain depth of space. At refineries, fugitive exhaust from on-site structures such as tanks, or a treatment facility may be modeled as a volume source. Release area, base elevation and area are needed for modeling. A roadway over which contaminated soil is hauled may also be modeled as a series of volume sources.

The important parameters used to characterize volume sources for dispersion modeling are location, release height and initial lateral and vertical dimensions. The AERMOD model user's guide has instructions on defining the initial lateral and vertical dimensions of the source. The length of the side of the volume source will need to be known, as will the vertical height of the source, and whether it is on or adjacent to a structure or building. The north-south and east-west dimensions of each volume source must be the same. For refined modeling, the location is simply expressed by a single east-west (X) and north-south (Y) coordinate.

Parameters

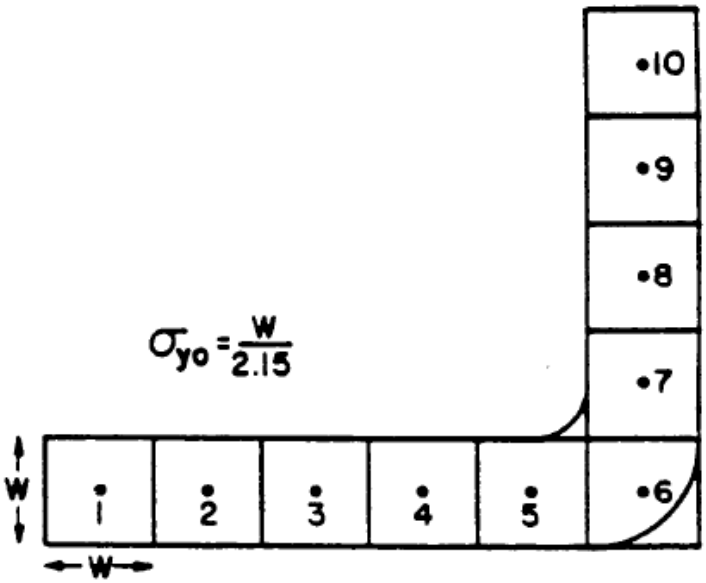
Emission rate: Enter the emission rate of the volume source.

Release height: Enter the height of the center of the volume source.

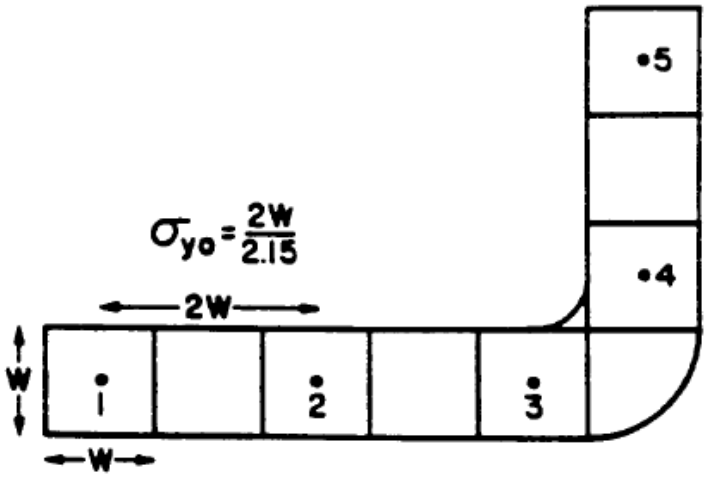
Initial lateral dimension: Enter the initial lateral dimension of the volume source. See "Volume Source Inputs" in the EPA's AERMOD User Guide for guidelines on estimating the initial lateral dimension of various types of volume and line sources. Use the following table to determine the initial lateral dimension.

Estimating Initial Lateral Dimension of Volume Sources

Type of Source	Formula
Single Volume Source	Length of side divided by 4.3
Line Source Represented by Adjacent Volume Sources	Length of side divided by 2.15
Line Source Represented by Separate Volume Sources	Center-to-center distance divided by 2.15



From U.S. EPA ISC Model Formulations Guide
A line source represented by adjacent volume sources



From U.S. EPA ISC Model Formulations Guide
A line source represented by separated volume sources

Initial vertical dimension: Enter the initial vertical dimension of the volume source. See "Volume Source Inputs" in the EPA's AERMOD User's Guide for guidelines on estimating the initial vertical dimension of various types of volume and line sources. Use the following table to determine the initial vertical dimension.

Estimating Initial Vertical Dimension of Volume Sources

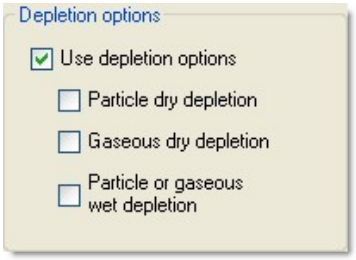
Type of Source	Formula
Surface-Based Source	Vertical dimension of source divided by 2.15
Elevated Source On or Adjacent to a Building	Building height divided by 2.15
Elevated Source Not On or Adjacent to a Building	Vertical dimension of source divided by 4.3

The open pit source type can be used to model particulate emissions (and gaseous emissions if the 09292 AERMOD executable is selected) from open pits, such as surface coal mines and rock quarries. The open pit algorithm in AERMOD uses an effective area for modeling pit emissions, based on meteorological conditions, and then utilizes the numerical integration area source algorithm to model the impact of emissions from the effective area sources.

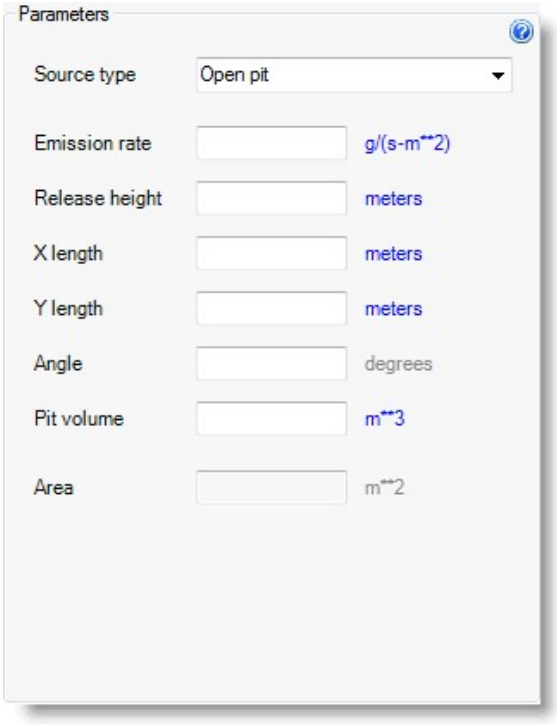
If the 07026 AERMOD executable is selected, all Open Pit sources must include particle deposition parameters even if the only result type selected is Concentration.

If the 09292 AERMOD executable is selected, Open Pit sources do require particle deposition parameters if:

- deposition is being modeled
- at least one source (it does not have to be an Open Pit source) specifies particle deposition parameters AND the Control Form options to explicitly turn off dry and wet depletion are not selected. The following image shows the depletion options in the Control option form all turned off.



Parameters



Emission rate: Enter the emission rate of the open pit source.

Release height: Enter the release height above the base of the pit. A release height of zero indicates the release is at the base of the pit.

X length: Enter the length of the X side of the open pit source (in the east-west direction if the rotation angle is 0 degrees).

Y length: Enter the length of the Y side of the open pit source (in the north-south direction if the rotation angle is 0 degrees).

Angle: (Optional) Enter the rotation or orientation angle for the open pit source in degrees from North, measured positive in the clockwise direction. If left blank, it is assumed that the open pit source is oriented in the north-south and east-west directions (i.e., the rotation angle is 0 degrees). If a non-zero value is entered in this field, the model will rotate the open pit area clockwise about the vertex. The vertex is defined by the X and Y coordinates that anchor the open pit source. The rotation angle may be positive (for clockwise rotation) or negative (for counterclockwise rotation), but the absolute value of the rotation angle cannot exceed 180 degrees.

Pit volume: Enter the volume of the open pit source

.

BREEZE Line

Line sources can be represented by using the Line Object Array tool.

The BREEZE enhanced version of AERMOD has a line source model object that simulates rectangular area sources.

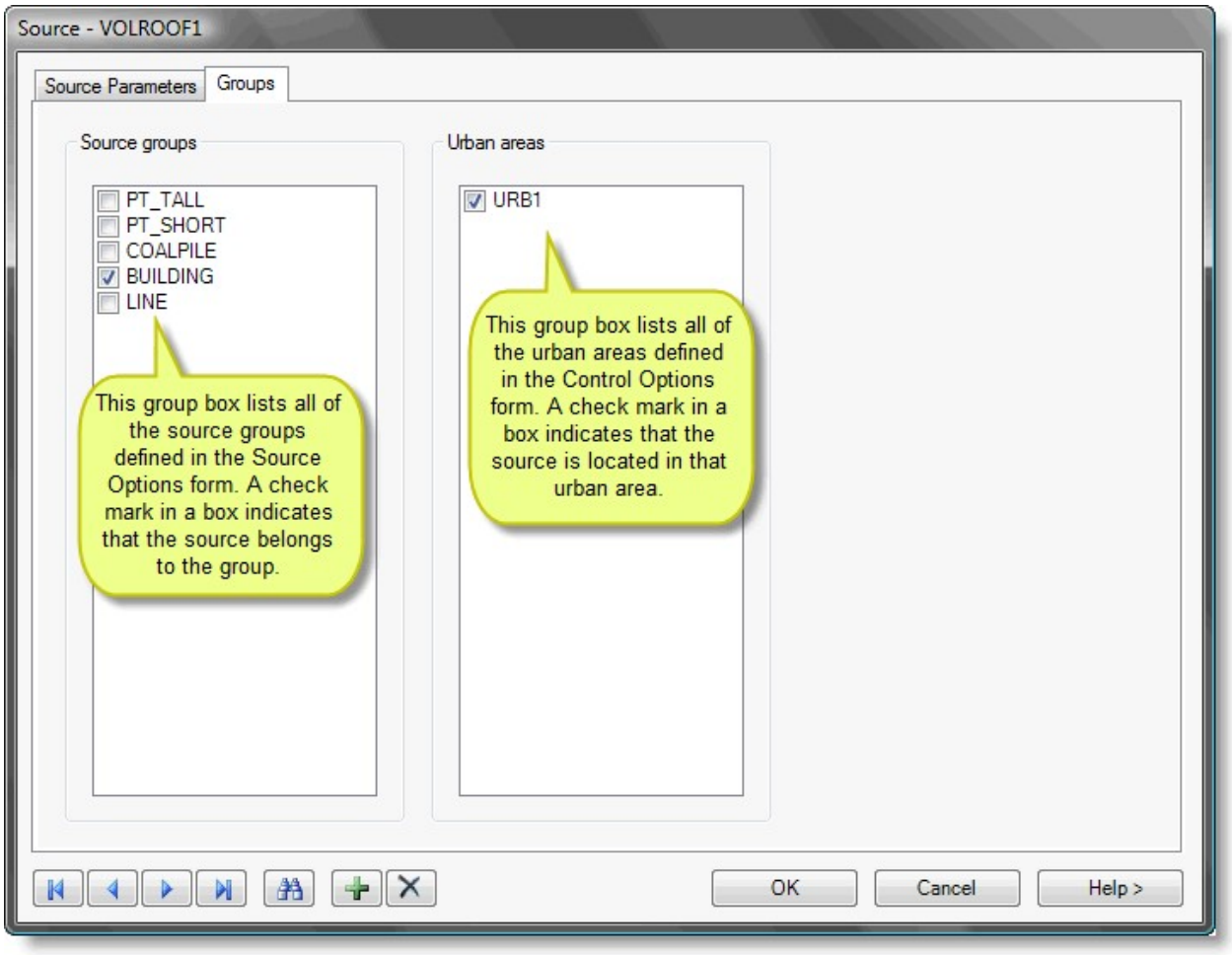
The BREEZE Enhanced Fortran executable must be selected for this source type to be used.

Source Enhancements

There are a number of source enhancements that can be made on a source by source basis. This section discusses those enhancements.

Groups

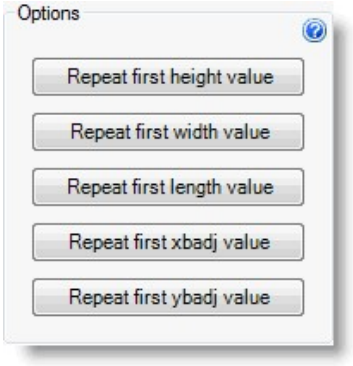
The groups tab displays the various groups that the source belongs two. There are two group types.


Source - VOLROOF1
Source Parameters
Groups
Source groups
PT_TALL
PT_SHORT
COALPILE
BUILDING
LINE
This group box lists all of the source groups defined in the Source Options form. A check mark in a box indicates that the source belongs to the group.
Urban areas
URB1
This group box lists all of the urban areas defined in the Control Options form. A check mark in a box indicates that the source is located in that urban area.
OK
Cancel
Help >

Building Downwash

Options

These buttons make it easy to change the parameters for a column. Pressing a button will replace all the cells in the column corresponding to the button with the value that is in the first cell of that column.



Building dimensions

These parameters are the results of the BPIP model run. They are only generated for Point type sources (Point, Capped Point, Horizontal Point, and Flare).

These parameters are typically not edited by the modeler.

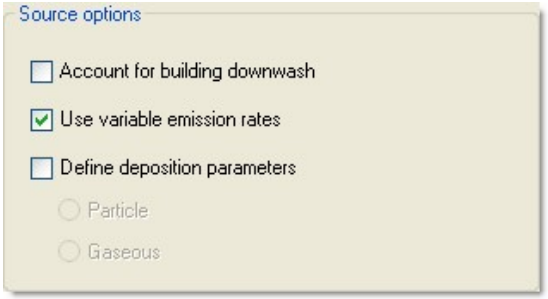
Building dimensions

	Height	Width	Length	XBadj	YBadj
10°	15.00	46.03	40.21	-84.13	-1.66
20°	15.00	51.01	46.48	-86.01	-12.75
30°	15.00	54.44	51.34	-85.27	-23.46
40°	15.00	56.22	54.64	-81.94	-33.45
50°	10.00	232.00	218.53	-219.65	-71.87
60°	10.00	244.90	198.97	-194.11	-88.17
70°	10.00	250.35	173.37	-162.68	-101.80
80°	10.00	248.20	142.50	-126.31	-112.33
90°	10.00	239.90	107.30	-86.10	-120.15
100°	0.00	0.00	0.00	0.00	0.00
110°	0.00	0.00	0.00	0.00	0.00
120°	0.00	0.00	0.00	0.00	0.00
130°	0.00	0.00	0.00	0.00	0.00
140°	10.00	218.53	232.00	-44.13	-110.38
150°	10.00	198.97	244.90	-34.28	-94.63
160°	10.00	173.37	250.35	-23.38	-76.00
170°	10.00	142.50	248.20	-11.77	-55.06
180°	10.00	107.30	239.90	0.20	-32.45
190°	10.00	116.95	240.29	10.13	-9.34
200°	10.00	153.65	236.29	17.84	12.15

Variable Emission Rates

See also Source Option Tools.

This tab is enabled / displayed when the "Use variable emission rates" check box is selected in the Source Parameters tab.

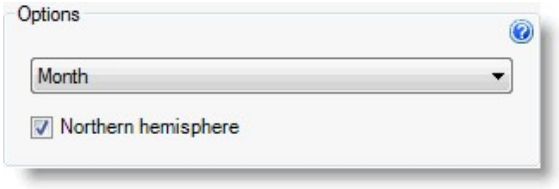


The model provides the option of specifying variable emission rate factors for individual sources. The factors may vary on different time scales, such as by season, hour-of-day, etc.

Emission rate factors are multiplied with the source's emission rate as specified in the Source Data tab.

You can also use hourly emissions files in the model. BREEZE has an Hourly Emission File Editor that can be used to create hourly emission files.

Options



Note that these emission factors are not necessarily regulatory default options and should be used only with approval from a reviewing regulatory authority. The use of these options for a permitting analysis would typically require enforceable permit conditions limiting operations to the conditions modeled.

Select from the drop down list box the temporal variation for this source.

Season - Model Keyword: SEASON - Option to specify an emission rate factor for each season. Enter the factor for each season. For example, winter includes December, January, and February. Total of 4 emission factors.

Month - Model Keyword: MONTH - Option to specify emission rate factors for each month. Total of 12 emission factors.

Hour of day - Model Keyword: HROFDY - Option to specify emission rate factors for each hour. Total of 24 emission factors.

Wind Speed - Model Keyword: WSPEED - Option to specify emission rate factors for wind speed categories as defined in the meteorological form. This factor category is typically used for emissions that are a function of wind erosion.

Season and hour of day - Model Keyword: SEASHR - Option to specify emission rate factors for each hour in each season. Total of 96 emission factors.

Season, hour of day and day of week - Model Keyword: SHRDOW - Option to specify emission rate factors for each hour for each weekday (Monday through Friday share the same emission factor), Saturday, and Sunday in each season. Total of 288 emission factors.

Season, hour of day and the seven days of the week - Model Keyword: SHRDOW7 - Option to specify emission rate factors for each hour for each weekday (Monday through Friday can have different emission factors), Saturday, and Sunday in each season. Total of 672 emission factors.

Month, hour of day, and day of week - Model Keyword: MHRDOW - Option to specify emission rate factors for each hour for each weekday (Monday through Friday share the same emission factor), Saturday, and Sunday in each month of the year. Total of 864 emission factors.

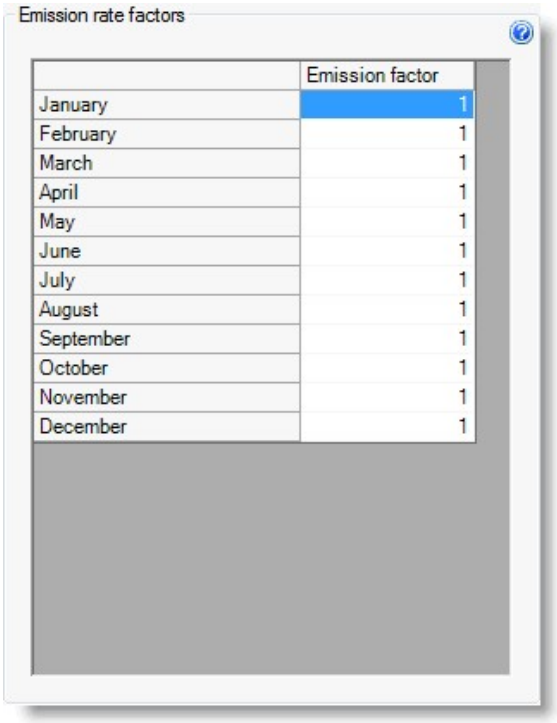
Month, hour of day, and the seven days of the week - Model Keyword: MHRDOW7 - Option to specify emission rate factors for each hour for each weekday (Monday through Friday can have different emission factors), Saturday, and Sunday in each month. Total of 2016 emission factors.

Hour of day, and day of week - Model Keyword: HRDOW - Available only when the 09292 AERMOD executable is selected. Option to specify emission rate factors that vary by hour-of-day, and day-of-week [M-F, Sat, Sun].

Hour of day, and seven days of the week - Model Keyword: HRDOW7 - Available only when the 09292 AERMOD executable is selected. Option to specify emission rate factors for each hour for each hour for each weekday (Monday through Friday can have different emission factors), Saturday, and Sunday.

Emission rate factors

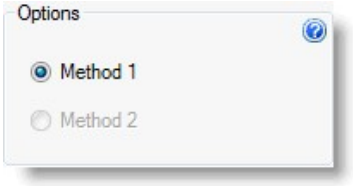
The number of cells associated with this field varies according to what Emission Factors option is selected. For each cell, enter the factor by which the emission rate is to be varied for the current source (e.g., 1 = 100 percent of the specified emission rate; 0.75 = 75 percent of the specified rate).



Particle Deposition

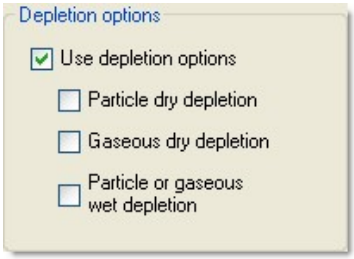
AERMOD includes algorithms to account for the gravitational settling and removal by dry and/or wet deposition of particulates. There are two methods of computing gravitational settling: Method 1 and Method 2.

Options



Please note that when using the 07026 AERMOD executable, if the Toxics option is NOT used, the modeler has the option to specify deposition parameters for a subset of modeled sources. These deposition parameters are always required when modeling Open Pit type sources or if the modeler wants to account for particle depletion from select sources. See Open Pit sources for more information.

When using the 09292 AERMOD executable, if ANY source has particle deposition data specified, ALL sources modeled will require particle deposition parameters IF dry or wet depletion is NOT turned off. Dry and wet depletion can explicitly be turned off when using the 09292 AERMOD executable in the Control Options form. The following image shows the depletion options in the Control Options form all turned off.



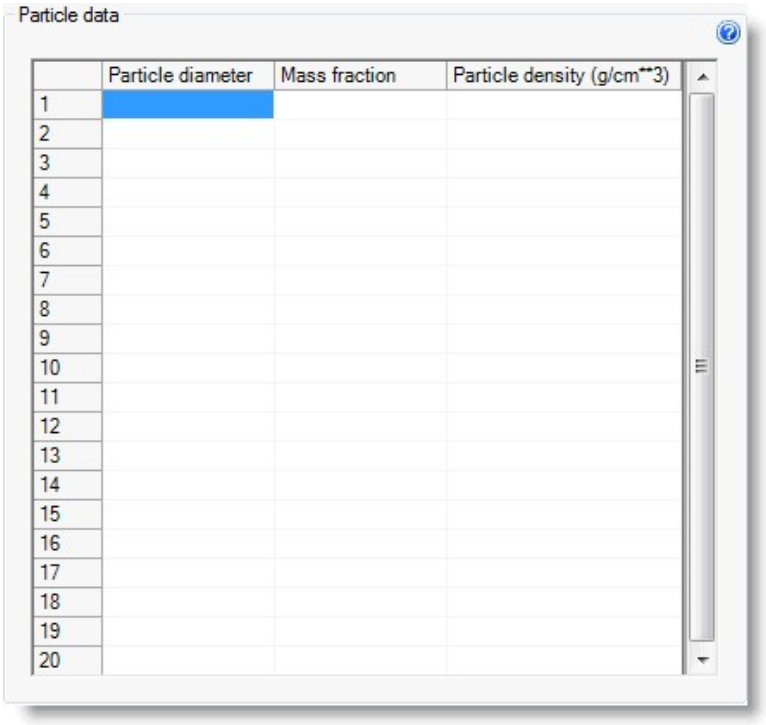
Method 1

- Method 1 can be applied if the regulatory default option is set or not in the Control Options form.
- Method 1 requires the user to enter pollutant particle size information specific to a particular source .

- Method 1 is used when a significant fraction (greater than about 10 percent) of the total particulate mass has a diameter of 10 µm or larger. The particle size distribution must be known reasonably well in order to use Method 1.
- This method must be used for Open Pit source types if the 07026 AERMOD executable is selected. If the 09292 AERMOD executable is selected, Method 2 may be used if the regulatory default option is not set in the Control Options form.

When Method 1 is selected, the following table is displayed. At a minimum, the first row of data must be provided.

Particle data (for Method 1)



Particle diameter: Enter the particle mass-mean diameter in microns.

Mass fraction: Enter the fraction (decimal) of the emission that is contained in each category (between 0 and 1). The total mass fraction must equal 1.00 (±0.02).

Particle Density: Enter the particle density in g/cm3.

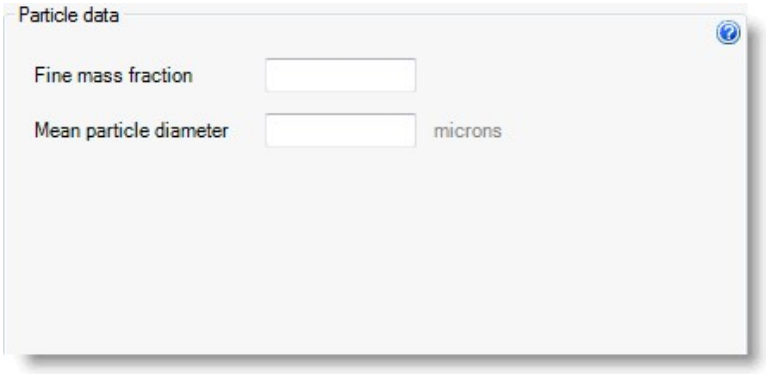
Note: You must specify particle information including particle diameter, mass fraction, and particle density when modeling an open pit source.

Method 2

This method is only available if the non-regulatory default option "Toxics" is enabled in the Dispersion/Averaging folder of the Control Options.

This method cannot be used for Open Pit source types.

Method 2 is used when the particle size distribution is not well known and when a small fraction (less than 10 percent of the mass) is in particles with a diameter of 10 µm or larger. The deposition velocity for Method 2 is given as the weighted average of the deposition velocity for particles in the fine mode (i.e., less than 2.5 µm in diameter) and the deposition velocity for the coarse mode (i.e., greater than 2.5 µm but less than 10 µm in diameter).



Fine mass fraction: Enter the fraction (decimal) of the emission that consists of particulate substance that is fine mode (smaller than 2.5 µm in diameter). See the appendix on "Particle Size Information for Selected HAPS – AERMOD-PRIME Method 1"

Mean particle diameter: Enter the particle mass-mean diameter in microns.

Gas Deposition

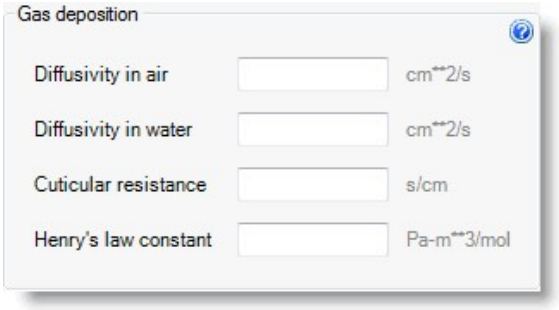
This set of optional input parameters are for dry and wet deposition of gaseous pollutants. They are only enabled when:

- (07026 AERMOD executable) the non-regulatory default "Toxics" option is selected in the the Control Options form.
- (09292 AERMOD executable) the "Use regulatory defaults" option is not checked in the Control Options form.

When using the 07026 AERMOD executable, these parameters should not be entered if the Seasonal and Land use gaseous deposition parameters in the Control Options form are not supplied.

Gas deposition

Model Keyword: GASDEPOS



Diffusivity in air: The diffusivity in air for the pollutant being modeled (cm^2/s).

Diffusivity in water: The diffusivity in water for the pollutant being modeled (cm^2/s).

Cuticular resistance: The cuticular resistance to uptake by lipids for individual leaves. See equation 25 in the appendix on "AERMOD-PRIME Deposition Theory".

Henry's law constant: Henry's Law constant ($Pa\text{-}m^3/mol$).

Urban Effects

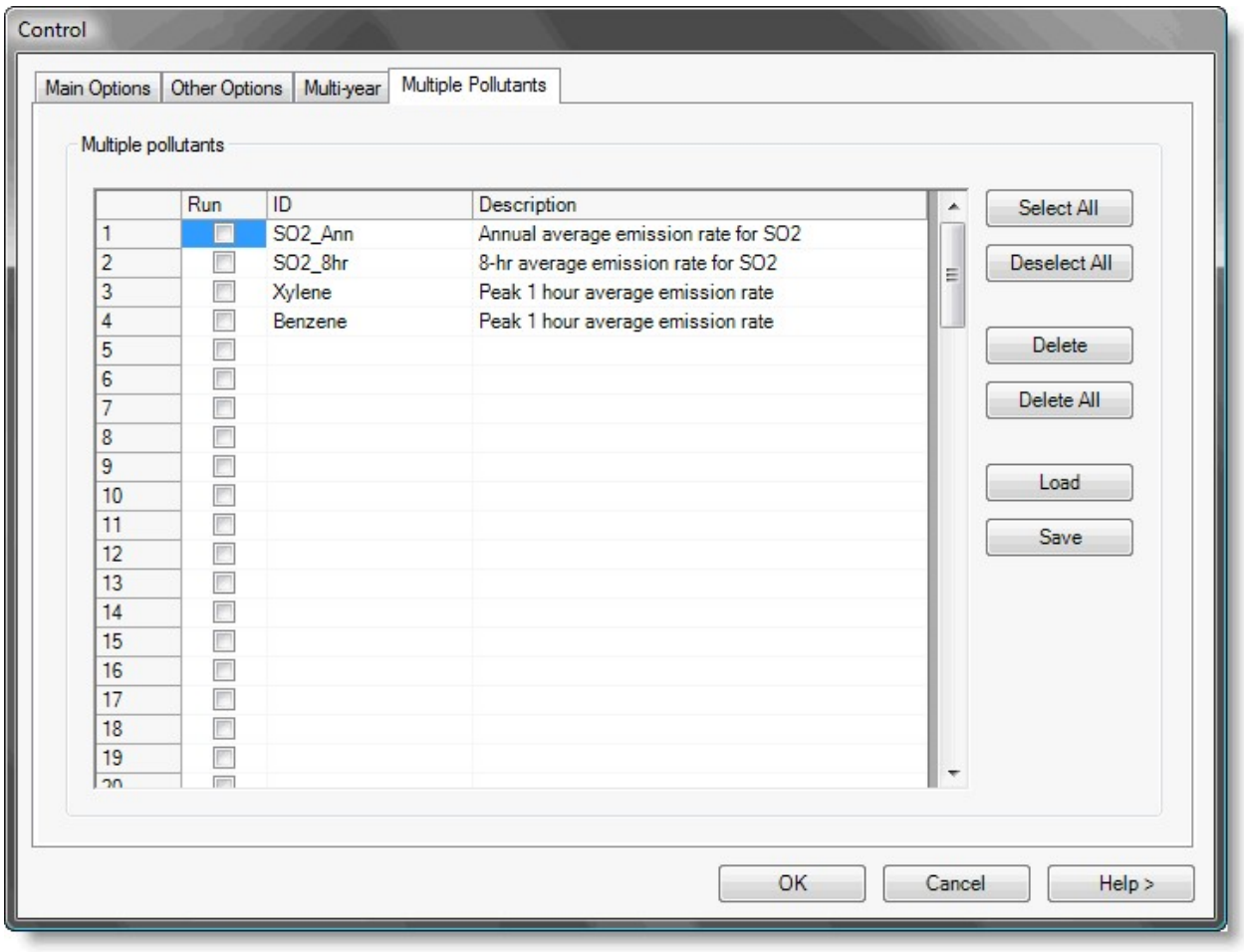
Urban effects can be specified for sources. To flag a source as belonging to a particular urban area, select the urban area from the Groups tab. Urban areas are defined in the Control Options form.

Multiple Emissions

AERMOD can only model a single pollutant or emission scenario in a single run unless the BREEZE MSP (Multi-Scenario Processor) version is used.

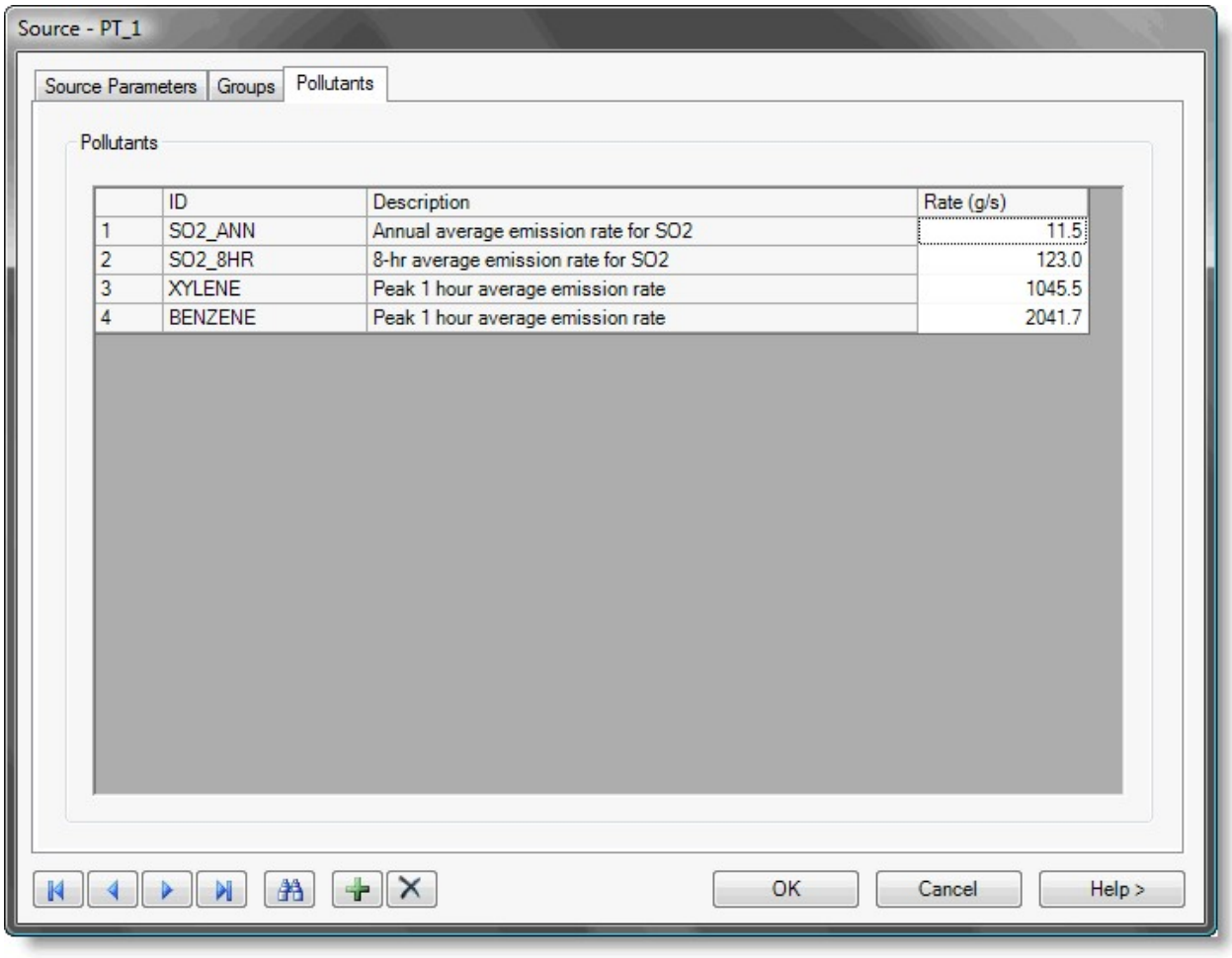
BREEZE is enhanced to allow the user to store multiple pollutants or emissions scenarios in a single project file. If your data set

includes more than one pollutant, choose the Pollutants tab from the Source screen to provide emission rates about a specific pollutant. The pollutants must first be entered in the Pollutants tab of the Control Form for this tab to be active.



Pollutants

This form list the pollutants identified in the Control Options form. For each source, enter the emission rate for each pollutant or emission scenario.



Notice that the emission rate field under the Source Parameters tab for each source is unavailable when the multiple pollutant option is selected.

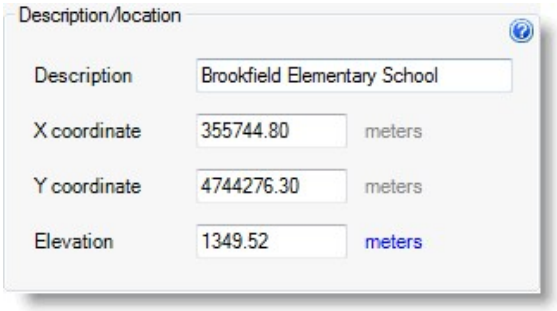
Receptor Types

Receptors are locations where concentration or deposition results are calculated.

Discrete

The Receptor dialog box provides a convenient method to enter and edit discrete receptor information. This dialog is automatically opened whenever you create a new discrete receptor or edit an existing receptor.

Location

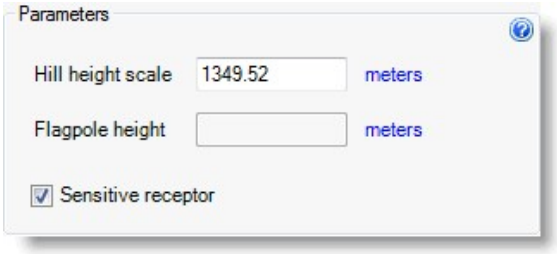


Description: Optional description to help identify the receptor.

X and Y coordinate: The X and Y coordinates of the receptor in meters.

Elevation (Z Coordinate): Enter the base height of the receptor above mean sea level.

Parameters



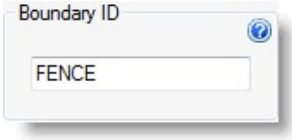
Hill height scale: The height scale computed by AERMAP to model terrain effects. This is not typically entered by the user but computed by AERMAP when importing terrain from a digital terrain file.

Flagpole height: The height of the receptor above ground level.

Boundary

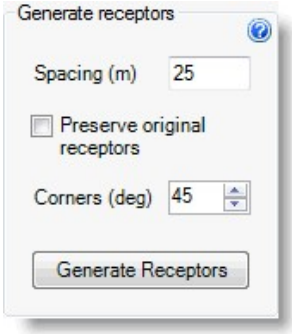
BREEZE allows you to create multiple boundary receptor groups. You may also specify receptor points along a boundary for modeling impacts at the boundary of the property. The Boundary Data dialog allows you to enter and edit boundary receptor information. This dialog is automatically opened whenever you create a new boundary or edit an existing boundary of receptors.

Boundary ID



Boundary ID: The boundary receptor group ID. Each time a new boundary receptor group is created, BREEZE will automatically assign and ID. Must be eight characters or less.

Generate receptors



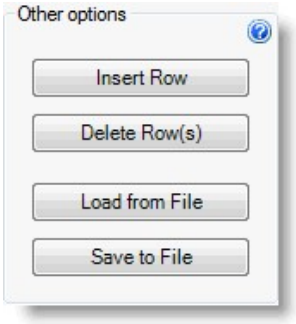
Spacing (m): The desired spacing, in meters, between receptors in the boundary receptor group.

Preserve original receptors: This option retains the original receptors when generating new receptors at a user defined interval. This will keep all of the original receptors in addition to adding new receptors at the user defined interval. This is useful for preserving the original boundary shape. This option is functionally the same as setting the corner tolerance to 0 degrees.

Corners (deg): The corner tolerance for receptor placement at the corners of the boundary receptor group. Receptors

Generate: Add receptors to the boundary separated by the specified receptor spacing distance

Other options



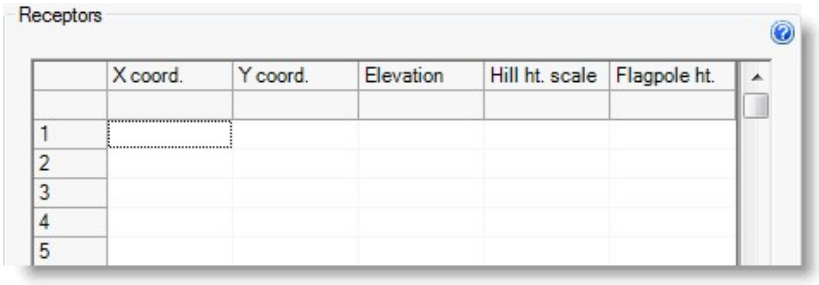
Insert Row: Allows you to add receptors to the boundary group.

Delete Row(s): Allows you to remove receptors from the boundary.

Load from File: Allows you to add receptors to the boundary group by loading an external file that contains receptor information.

Save to File: Allows you to save receptors to an external file.

Receptors



X coord.: The X coordinate of the receptor.

Y coord.: The Y coordinate of the receptor.

Elevation: The ground elevation (above mean sea level) of the receptor in meters. This value is normally obtained during the importation of terrain data.

Hill ht. scale: The computed height scale for the receptor used in AERMOD to calculated the dividing streamline height. This value is computed by AERMAP during the importation of elevation data. This value should never be less than the elevation.

Flagpole ht.: The height above ground level (AGL) for the receptor.

Grid

Two types of receptor networks are allowed by the model. A Cartesian grid network includes an array of points identified by their X (east - west) and Y (north - south) coordinates. A polar network is an array of points identified by direction and distance from a user-defined origin. Each of these grid types has a series of parameters associated with it that are used to define the network, including any receptor elevations for elevated terrain and flagpole receptor heights.

The Gridded Receptor Network dialog allows you to enter and edit gridded receptor networks in the current project. Users have control over the type of network used, the terrain elevations, and flagpole receptor heights. Data can be manually entered for the terrain elevation or receptor height information. Automation of terrain importation using digital elevation models (DEM's) can be performed using the Tools/Import terrain menu item. Gridded receptors are typically used to compute pollutant impacts over a large area and graphically illustrate contours of pollutant impacts.

You can specify the following grid types on the Grid form:

Uniform Cartesian Grid

Non-uniform Cartesian Grid

Uniform Polar Grid

Non-uniform Polar Grid

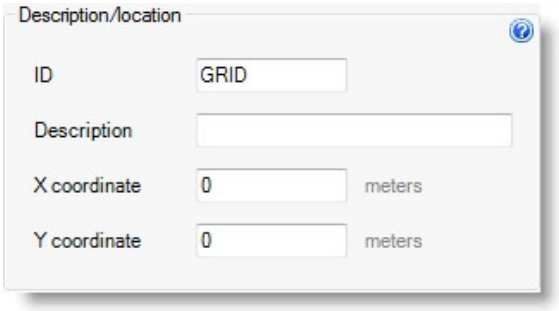
3D Cartesian Grid

Once you have specified a grid, you can adjust the location, size, and spacing of the grid, enter or edit elevations in the Terrain Elevations folder, specify flagpole receptor heights in the Flagpole Receptor Heights folder, or if using AERMOD, specify Height Scales.

Grid Parameters

The Grid Parameters allows you to specify basic receptor grid configuration information.

Description/location



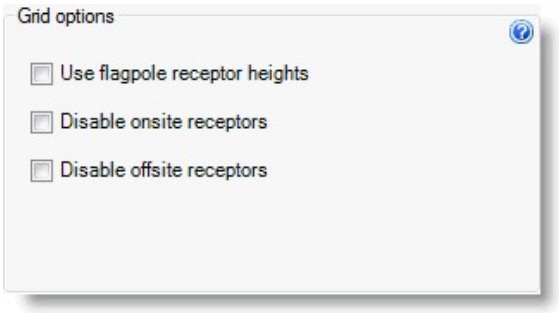
ID: Enter a unique alphanumeric identifier for each receptor grid, up to eight characters. Each grid ID should contain at least one alphabetical character. BREEZE will automatically assign an ID when a grid is created.

Note: Do not use a hyphen (-) or space when specifying a grid ID (e.g., do not enter GRID-1). If you wish to use a separator, type an underscore "_" (e.g., GRID_1).

Description: Enter an optional description with additional grid information, up to thirty characters.

X and Y Coordinate: Enter the X and Y coordinates of the grid in meters. For Cartesian grids, X (East-West) and Y (North-South) are coordinates in meters of the southwest corner of the grid. For polar grids, X (East-West) and Y (North-South) are coordinates in meters of the center of the grid.

Grid options



Use flagpole receptor heights: Select this option to enter receptor heights above local terrain (i.e., flagpole receptor heights). When selected, the Flagpole Receptor Heights folder appears.

Disable onsite receptors: Select this option to remove grid receptors within the bounds of a boundary receptor.

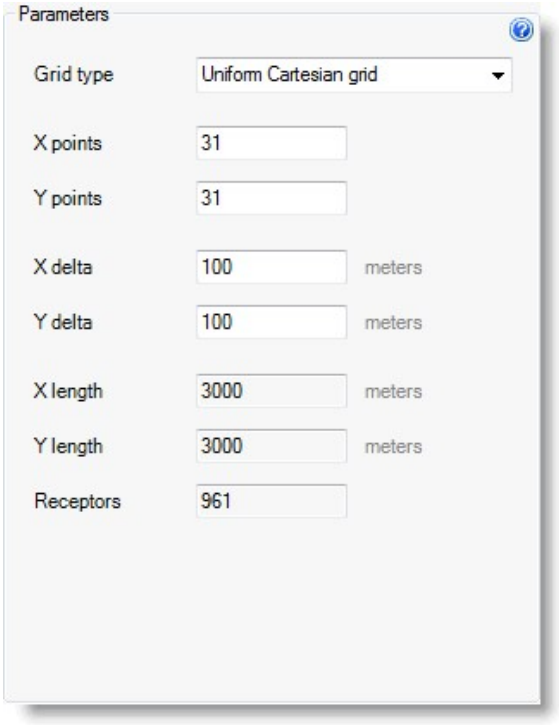
Disable offsite receptors: Select this option to remove grid receptors outside the bounds of a boundary receptor.

Grid Types

There are four types of receptor grids that can be created. Each have slightly different parameters that are used to define the extent and location of the receptors within the grid.

1. Uniform Cartesian
2. Non-uniform Cartesian
3. Uniform polar
4. Non-uniform polar
5. 3D Cartesian

Uniform Cartesian



X points: The number of receptors on the X (East-West) axis of the grid.

Y points: The number of receptors on the Y (North-South) axis of the grid.

X delta: The spacing, in meters, between each receptor on the X (East-West) axis of the grid.

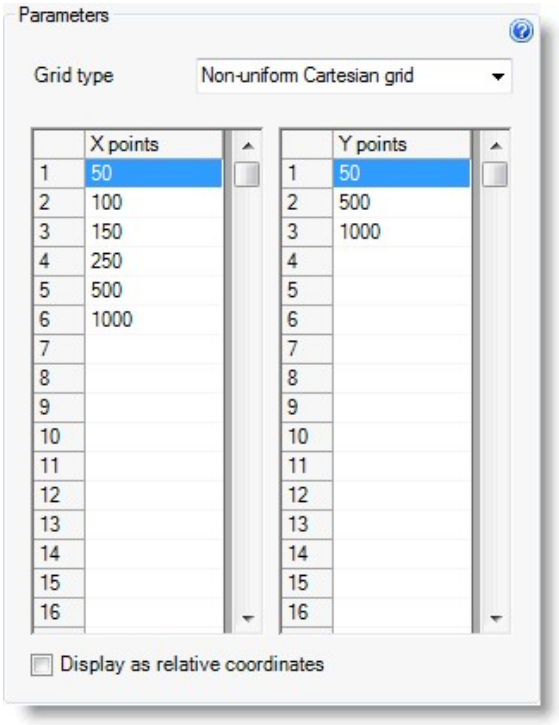
Y delta: The spacing, in meters, between each receptor on the Y (North-South) axis of the grid.

X length: The total distance in the X direction of the receptor grid.

Y length: The total distance in the Y direction of the receptor grid.

Once you have specified a Uniform Cartesian grid, you can enter Terrain Elevations and Flagpole Receptor Heights of each receptor.

Non-uniform Cartesian

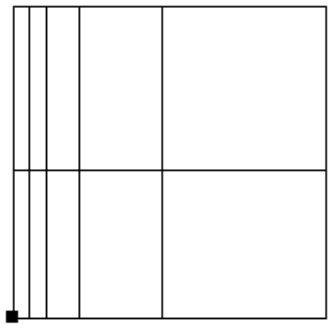


X points: Enter the X coordinates of the non-uniform Cartesian grid. Each coordinate entered represents a series of receptor locations on the Non-uniform Cartesian grid in the X axis.

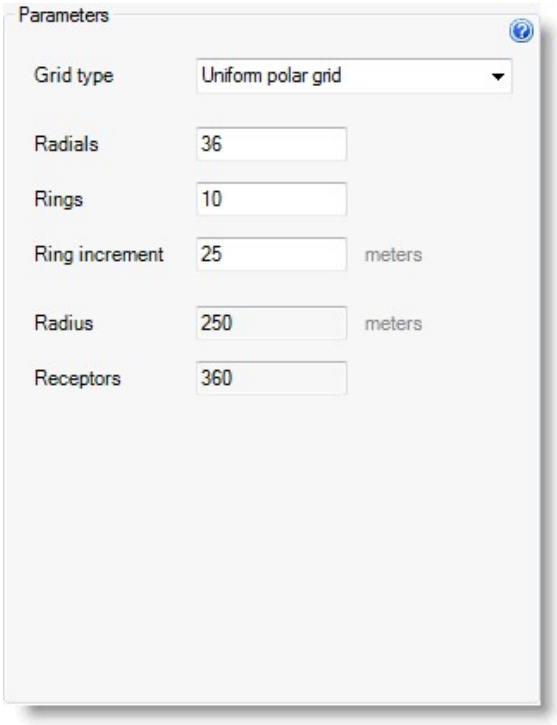
Y points: Enter the Y coordinates of the non-uniform Cartesian grid. Each coordinate entered represents a series of receptor locations on the Non-uniform Cartesian grid in the Y axis.

Display as relative coordinates: The X and Y points are displayed as distances relative to the X and Y coordinates.

The above parameters result in a receptor grid as displayed below:



Uniform polar



Radials: Enter the number of radials. The first radial is anchored relative to North at 0 degrees. For example, BREEZE will produce radials at 0, 90, 180, and 270 degrees if you set the number of radials to 4.

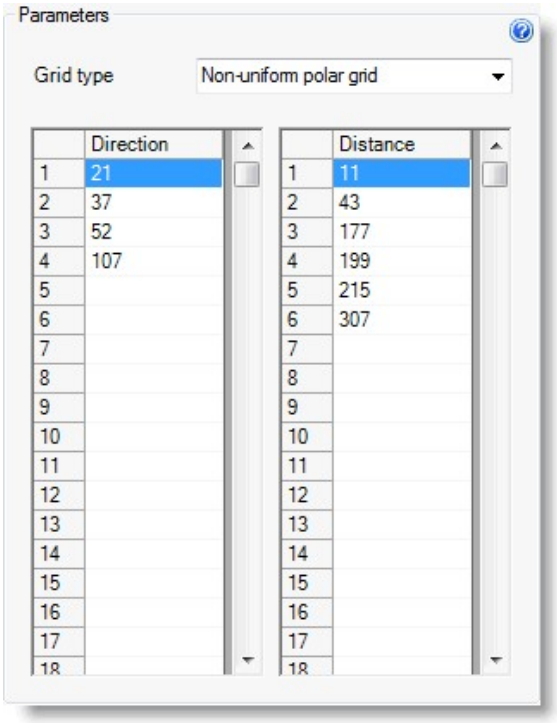
Rings: Enter the number of rings to generate from the center of the grid.

Ring Increment: Enter the distance between rings in meters.

Radius: The product of the number of rings and the ring increment. This parameter can not be edited.

Receptors: The total number of receptors for this grid. This parameter can not be edited.

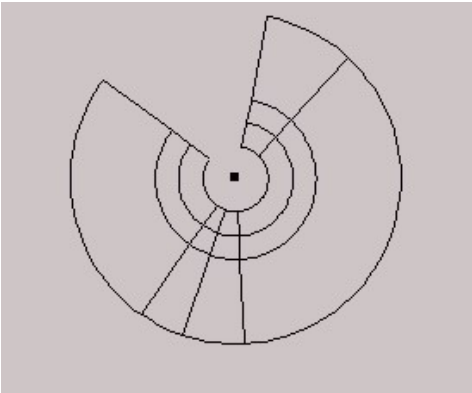
Non-uniform polar



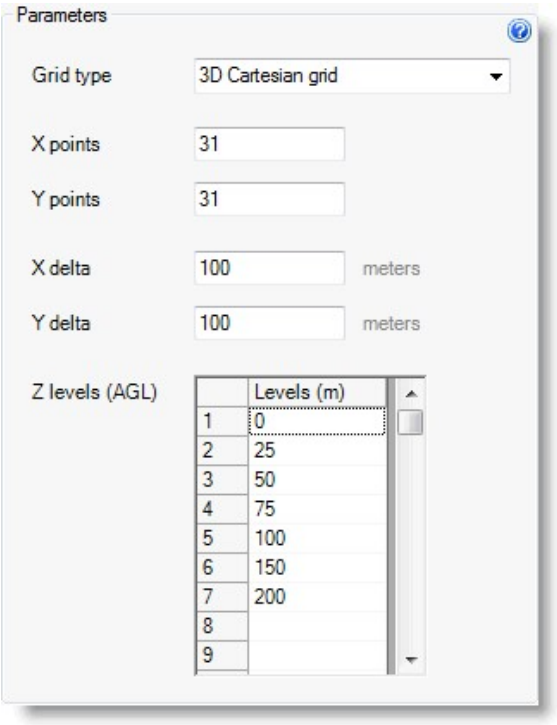
Direction: Enter each direction radial for the non-uniform polar grid.

Distance: Enter the distance of each ring from the center of the non-uniform polar grid.

The below image represents a non-uniform polar grid with the direction and distances specified in the above image.



3D Cartesian



X points: The number of receptors on the X (East-West) axis of the grid.

Y points: The number of receptors on the Y (North-South) axis of the grid.

X delta: The spacing, in meters, between each receptor on the X (East-West) axis of the grid.

Y delta: The spacing, in meters, between each receptor on the Y (North-South) axis of the grid.

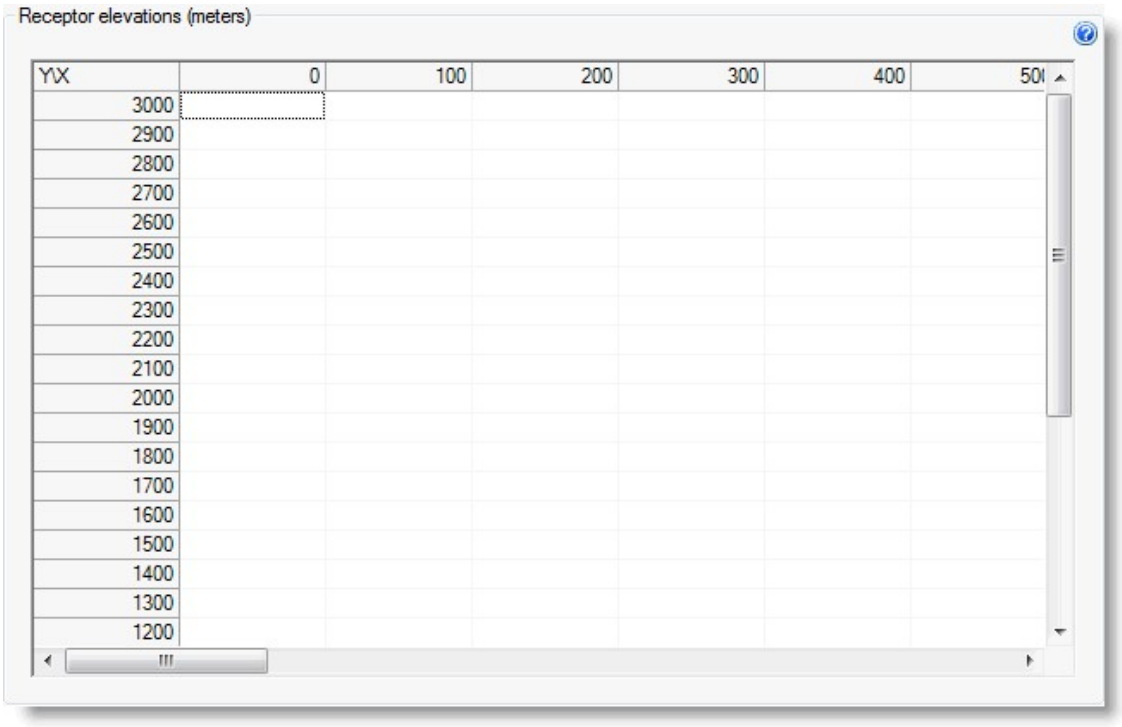
Z levels (AGL): The heights above ground level (AGL) in meters for each 2-dimensional grid.

Receptor Elevations

Terrain elevations are most easily entered from USGS or Ordnance Survey DEM (Digital Elevation Model) files as described under "Importing Terrain Elevations" You may also manually enter terrain elevation data. To do this, select the Terrain Elevations folder to enter terrain elevations for each receptor on the Terrain Elevations table.

Receptor elevations

Each cell on this form can be edited. For a Cartesian grid, the X coordinates are displayed across the top of the grid and the Y coordinates displayed down the left. For a polar grid, the ring distances are displayed across the top of the grid and the radial degrees are displayed down the left.

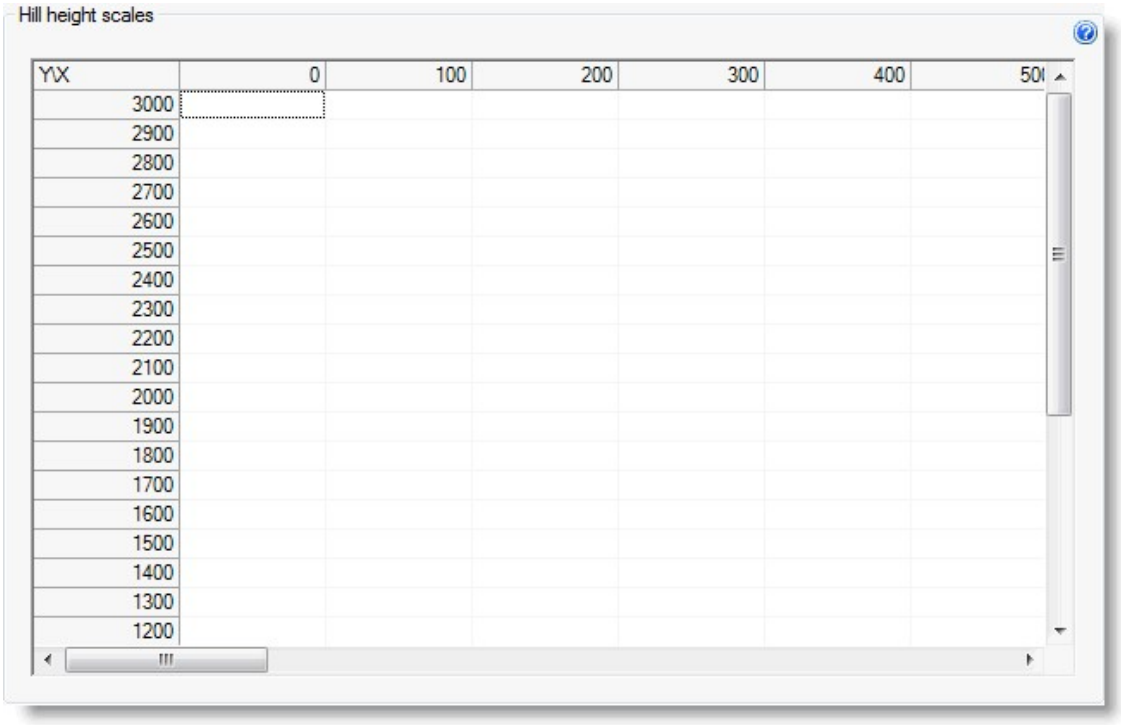


You can cut (cntl+x), copy (cntl+c), paste (cntl+v) or delete (del) data in the cells. You can also copy and paste data from existing spreadsheets.

Hill Height Scales

Hill height scales

This folder typically is not manually edited. AERMAP (accessible from the Import Terrain from the Tools menu) calculates these values. You can select the Hill Height Scales folder to edit/view receptor height scales. Each cell in this folder can be edited. Edit the scale height for each flagpole receptor. For a Cartesian grid, the receptor X coordinates are displayed across the top of the screen and the Y coordinates are displayed on left. For a polar grid, the ring distances are displayed across the top of the screen and the radial degrees are displayed on the left.

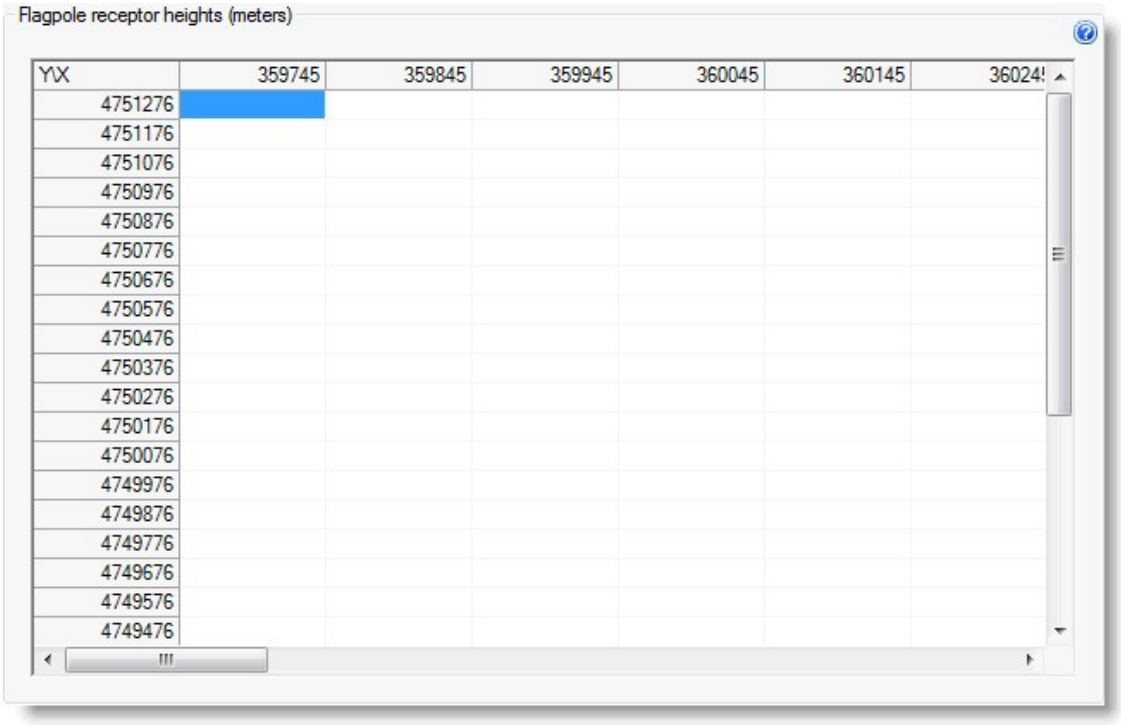


You can cut (cntl+x), copy (cntl+c), paste (cntl+v) or delete (del) data in the cells. You can also copy and paste data from existing spreadsheets.

Flagpole Receptor Heights

Flagpole receptor heights

Select the Flagpole Receptor Heights folder to enter receptor heights above local terrain (i.e., flagpole receptor heights). The Receptor Heights table appears. Each cell on this form can be edited. Enter the height above ground for each flagpole receptor in meters. For a Cartesian grid, the receptor X coordinates are displayed across the top of the screen and the Y coordinates are displayed on left. For a polar grid, the ring distances are displayed across the top of the screen and the radial degrees are displayed on the left.

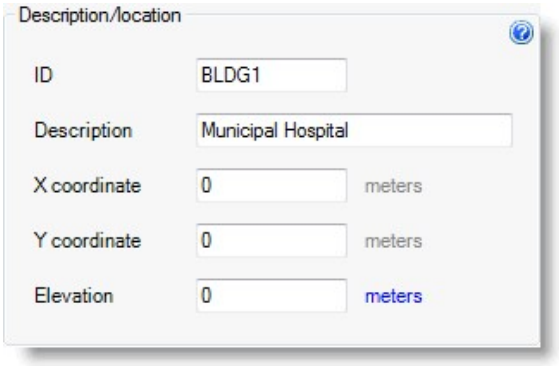


You can cut (cntl+x), copy (cntl+c), paste (cntl+v) or delete (del) data in the cells. You can also copy and paste data from existing spreadsheets.

Buildings

The Building form allows you to choose the building type (rectangular building, circular building, polygon building) and edit building parameters. The Building form is automatically opened whenever you create a new building.

Description/location



ID: Enter a unique alphanumeric identifier for each building, up to eight characters. Each building ID should contain at least one alphabetical character.

Description: Enter a brief description or a title to the building (optional).

X coordinates: The X coordinate of the building handle. This field is automatically populated when the building is first imported or drawn in the project. By changing the data in this field, you can change the origin of the building.

Y coordinates: The Y coordinate of the building handle. This field is automatically populated when the building is first imported or drawn in the project. By changing the data in this field, you can change the origin of the building.

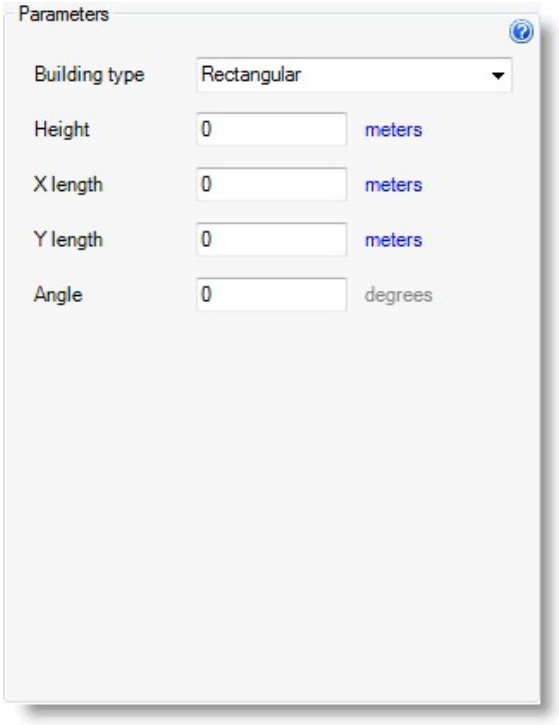
Elevation: Enter the elevation of the building. The default elevation is 0, but can be updated by using the Import Elevations tool.

Note: Buildings only need to be entered if Point type sources (Point, Capped Point, Horizontal Point, Flare) are modeled.

Note: The Building Profile Input Program (BPIP) determines the appropriate downwash parameters for each point type source.

Rectangular

Rectangular Building Parameters



Height: Enter the height of the rectangular building above ground level.

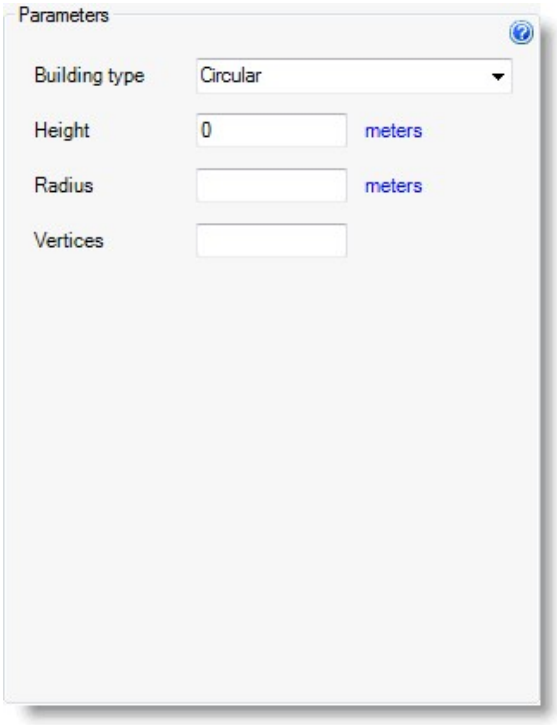
X length: Enter the length of the X side of the building (in the east-west direction if the rotation angle is 0 degrees).

Y length: Enter the length of the Y side of the building (in the north-south direction if the rotation angle is 0 degrees).

Angle: (Optional) Enter the rotation or orientation angle for the building in degrees from North, measured positive in the clockwise direction. If left blank, it is assumed that the building is oriented in the north-south and east-west directions (i.e., the rotation angle is 0 degrees). If a non-zero value is entered in this field, the model will rotate the building clockwise about the vertex. The vertex is defined by the X and Y coordinates that anchor the building (i.e., the building handle). The rotation angle may be positive (for clockwise rotation), or negative (for counterclockwise rotation), but the absolute value of the rotation angle cannot exceed 180 degrees.

Circular

Circular Building Parameters



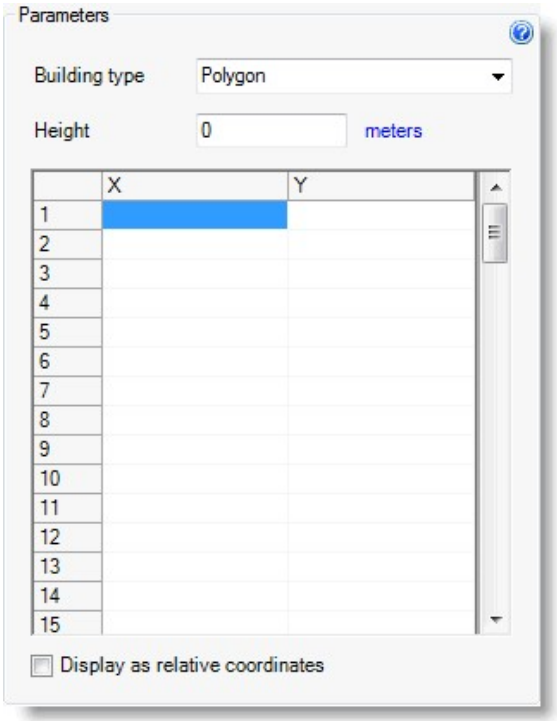
Height: Enter the height of the circular building above ground level.

Radius: Enter the radius of the circular building. This field is automatically filled when the circular building is first drawn in the project. By changing the number in this field, you can re-size the circular building.

Vertices: The number of vertices that make up the circular sources.

Polygon

Polygon Building Parameters

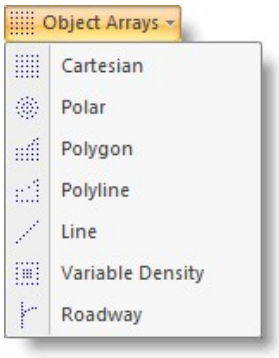


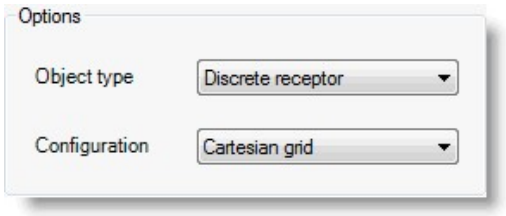
Height: Enter the height of the building above ground level.

X and Y: Enter the absolute X and Y coordinates for each vertex (corner) of the polygon building. The first vertex must always have the same X and Y coordinate as the object handle.

Object Arrays

BREEZE gives you the unique ability to create arrays of selected model object types. This is a convenient way to create a series of model objects quickly and uniformly.





Object type: Select the type of object array you would like to create (Source, Discrete receptor, or Building)

Configuration: Select the array configuration the model objects are to be created in.

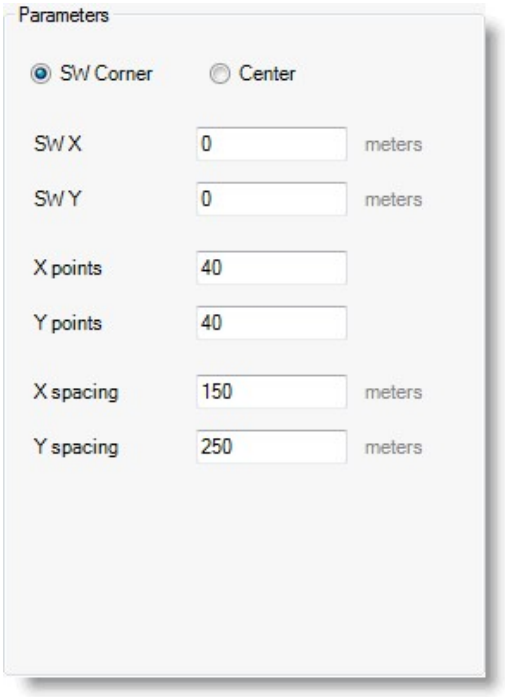
- Cartesian grid
- Polar grid
- Polygon grid
- Polyline
- Line
- Variable density
- Roadway

Cartesian Grid

A Cartesian grid configuration results in an array of model objects uniformly spaced in both the X and Y directions.

An example of a Cartesian grid model object array is provided here.

Parameters - Cartesian Grid



SW X (SW Corner): The Southwest X coordinate that defines the origin of the model object array.

SW Y (SW Corner): The Southwest Y coordinate that defines the origin of the model object array.

X (Center): The center X coordinate that defines the origin of the model object array.

Y (Center): The center Y coordinate that defines the origin of the model object array.

X points: The number of model objects to create in the X direction.

Y points: The number of model objects to create in the Y direction.

X spacing: The spacing between model object centers in the X direction.

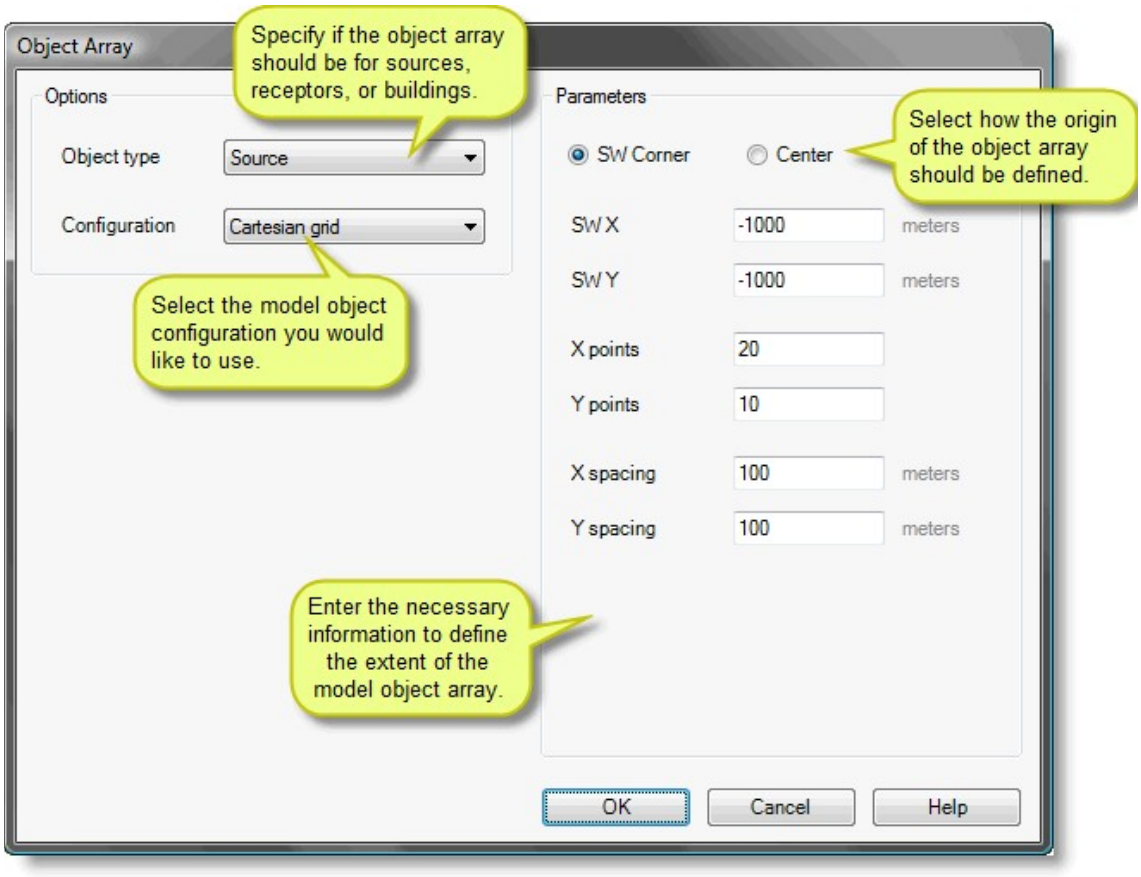
Y spacing: The spacing between model object centers in the Y direction.

Example

The following is an example of a Cartesian grid made up of rectangular area sources.

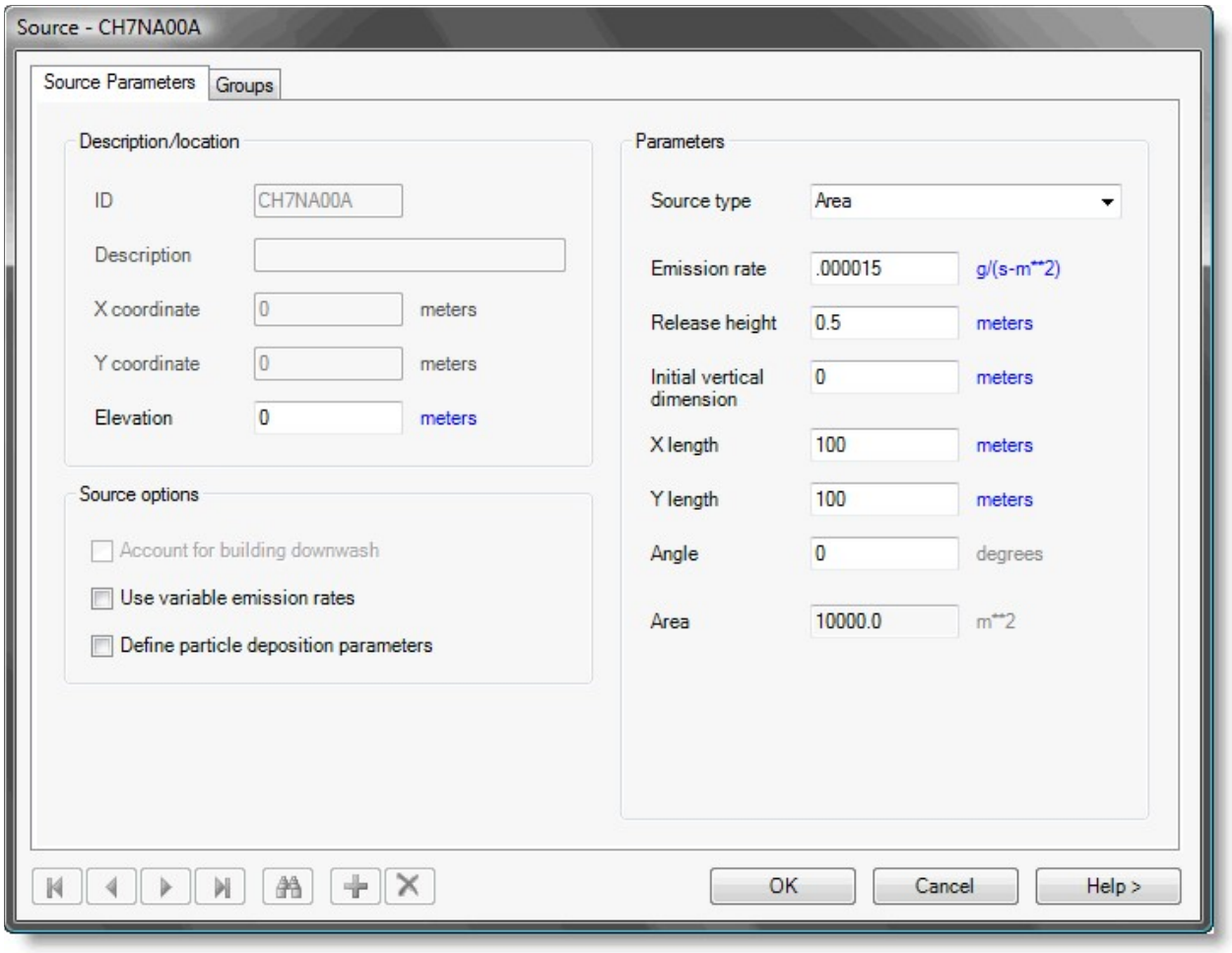
Select the Object Array Configuration

1. Select Cartesian from Drawing... | Object Arrays menu on the Map tab.
2. Draw the region you would like to have the model objects cover.
3. The following form appears.
4. Select the Object type (.i.e., Source).
5. Click the OK button.



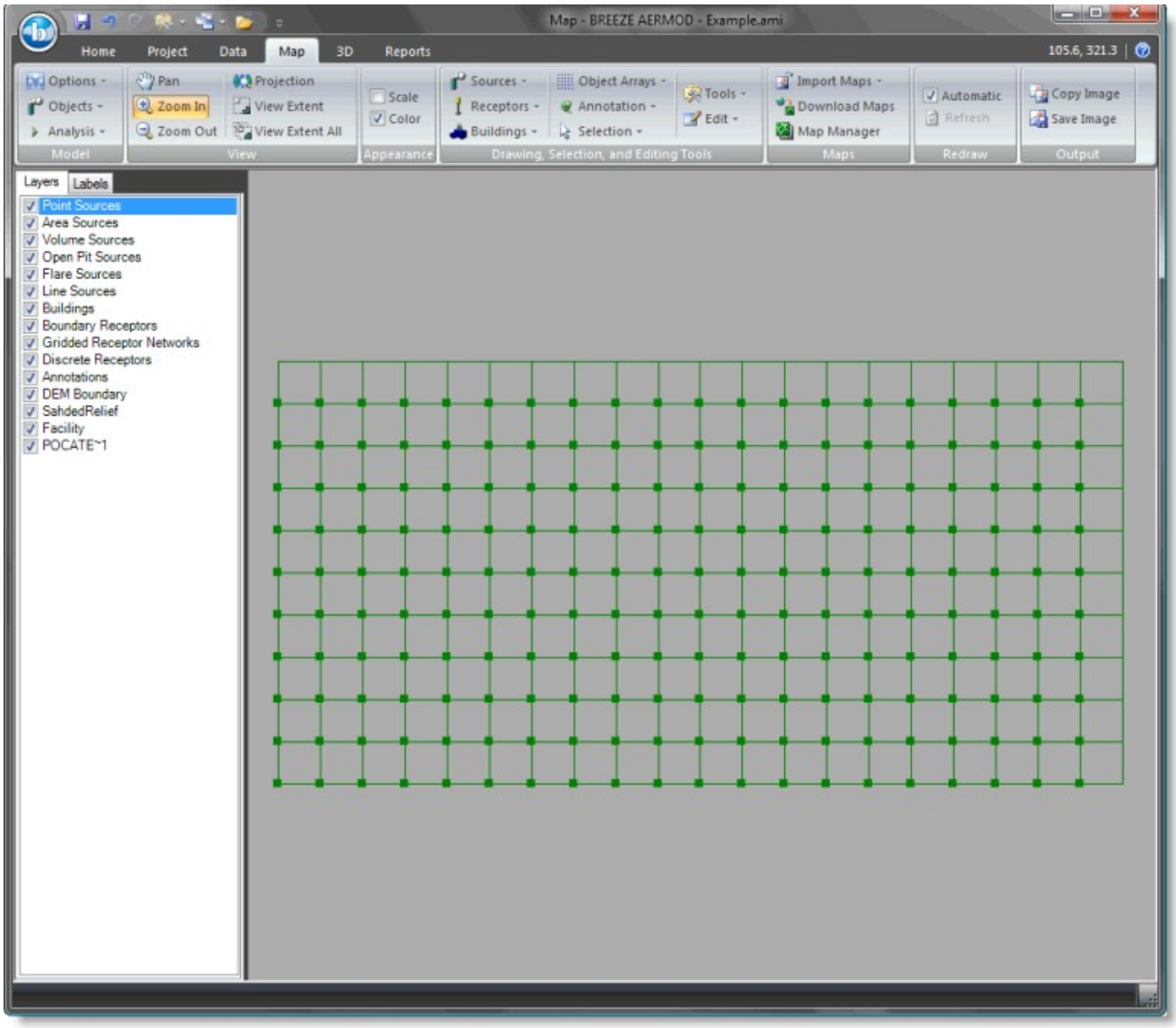
Specify Model Object Parameters

1. Enter the model object parameters. In this example, a Rectangular Area Source was select as the model object.
2. Click the OK button.



View Model Objects

This example results in a 20 x 10 array of rectangular area sources.

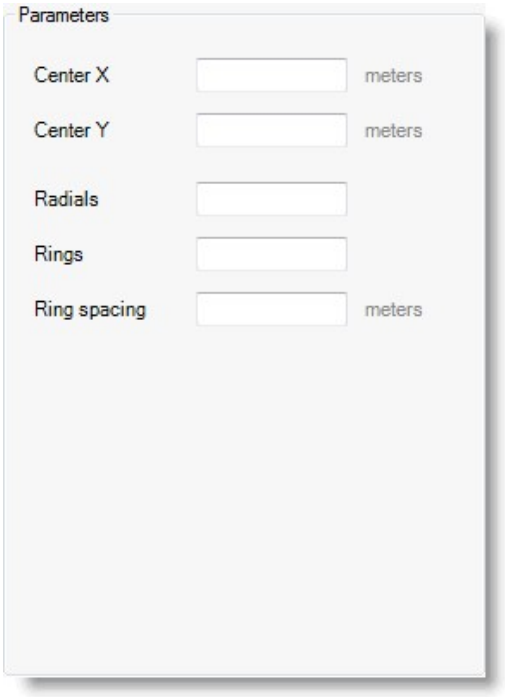


Polar

A Polar grid configuration results in an array of model objects distributed about the center of a polar grid.

An example of a Polar grid model object array is provided here.

Parameters - Polar Grid



X (center): The center X coordinate that defines the origin of the model object array.

Y (center): The center Y coordinate that defines the origin of the model object array.

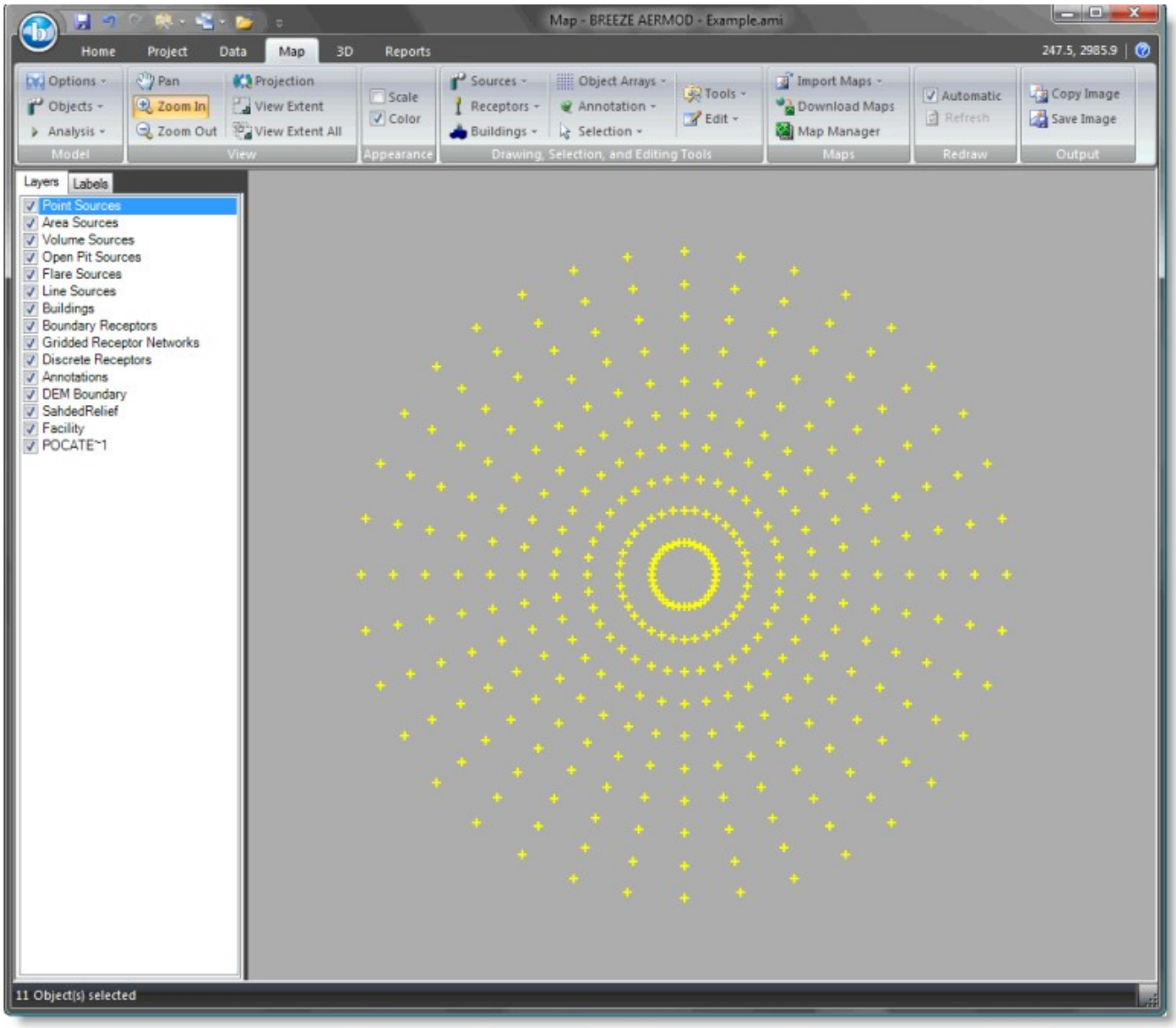
Radials: The number of radials extending from the polar grid origin.

Rings: The number of rings

Ring spacing: The spacing between each ring.

Example

The following is an example of a Polar grid discrete receptor model object array.

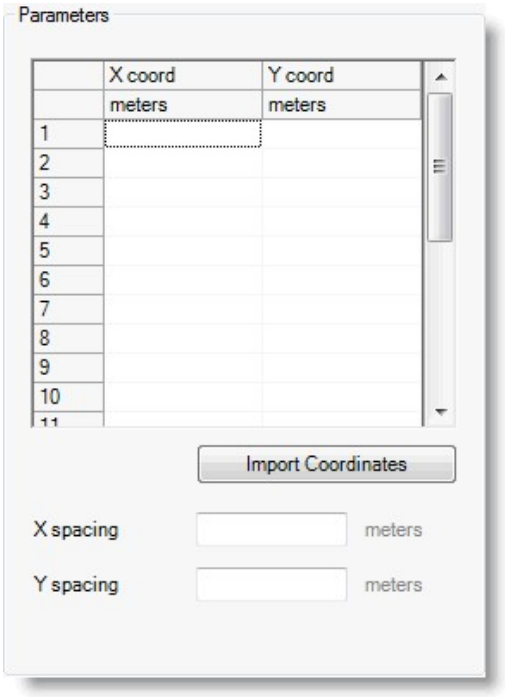


Polygon Grid

A Polygon grid configuration results in an array of model objects distributed throughout a user defined polygon.

An example of a Polygon grid model object array is provided here.

Parameters - Polygon Grid



X coord: The X coordinate of a corner of the polygon.

Y coord: The Y coordinate of a corner of the polygon.

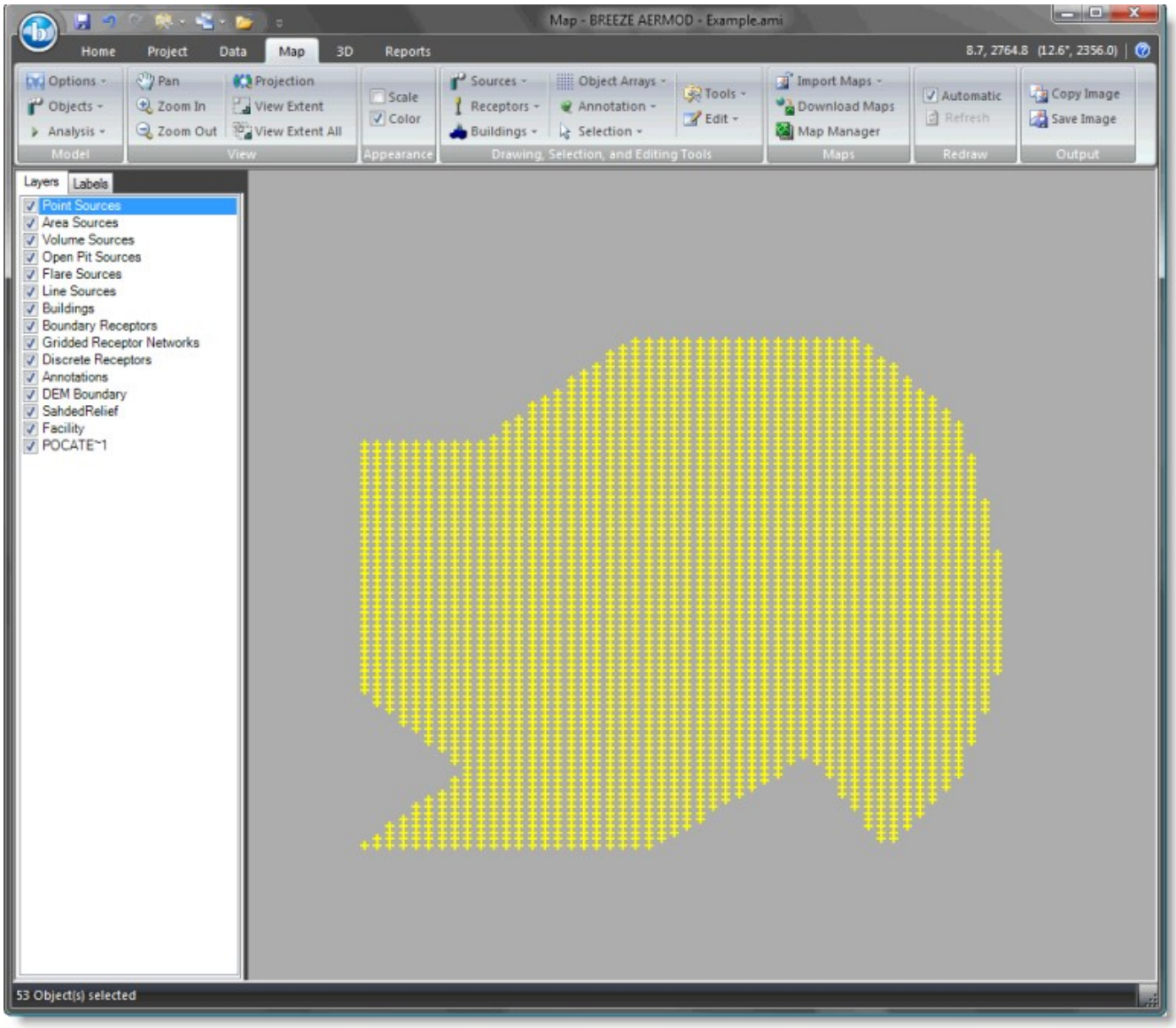
X spacing: The spacing between the center of each model object along the X axis.

Y spacing: The spacing between the center of each model object along the Y axis.

Import Coordinates: Option to import coordinates from an external .bln file.

Example

The following is an example of a Polygon grid discrete receptor model object array.

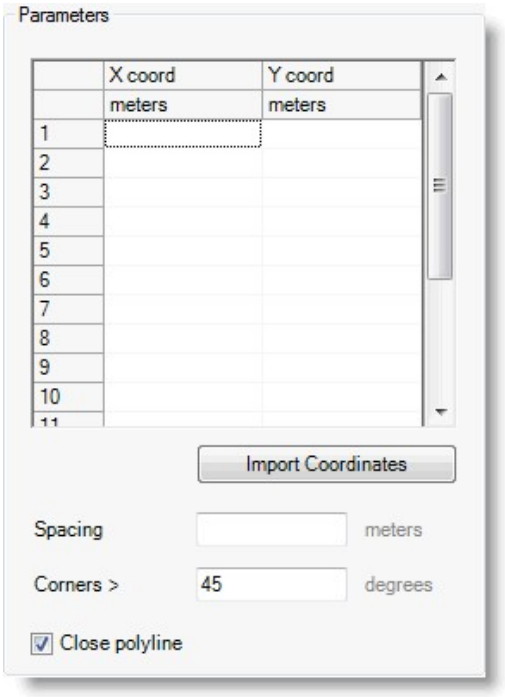


Polyline

A Polyline configuration results in an array of model objects distributed along a user defined polyline.

An example of a Polyline model object array is provided here.

Parameters - Polyline



X coord: The X coordinate of a corner of the polyline.

Y coord: The Y coordinate of a corner of the polyline.

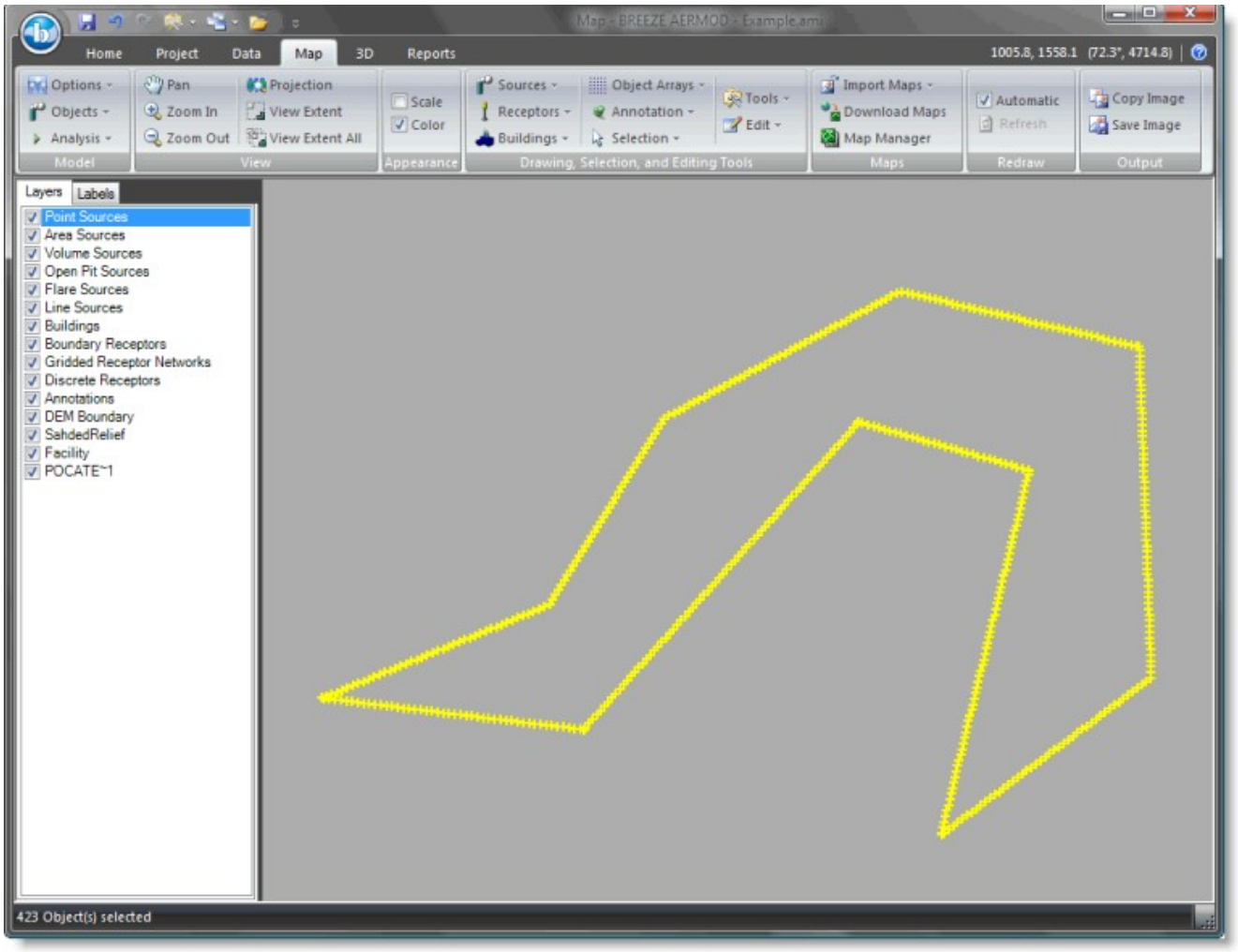
Spacing: The spacing between each model object center along the polyline.

Corners >: Corners along the polyline that are less than this value will have a model object placed at that corner.

Import Coordinates: Option to import coordinates from an external .bln file.

Example

The following is an example of a Polyline discrete receptor model object array.

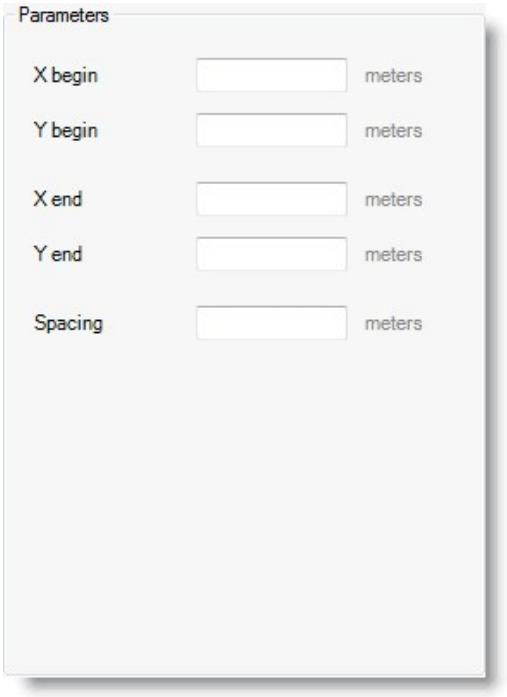


Line

A Line configuration results in an array of model objects distributed along a user defined line.

An example of a Line model object array is provided here.

Parameters - Line



X begin: The starting X coordinate of the line.

Y begin: The starting Y coordinate of the line.

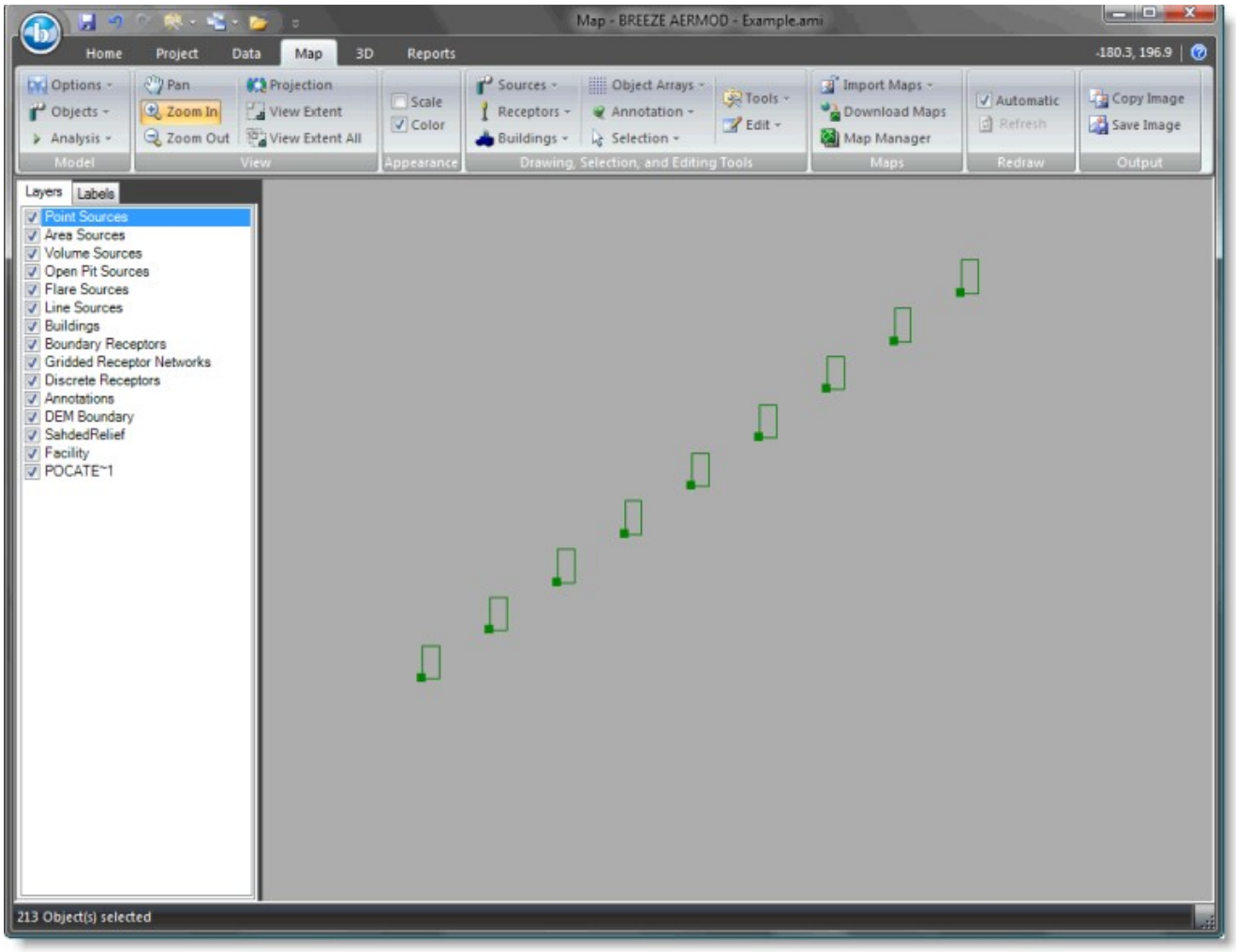
X end: The ending X coordinate of the line.

Y end: The ending Y coordinate of the line.

Spacing: The spacing between each model object center along the line.

Example

The following is an example of a Line area source model object array.

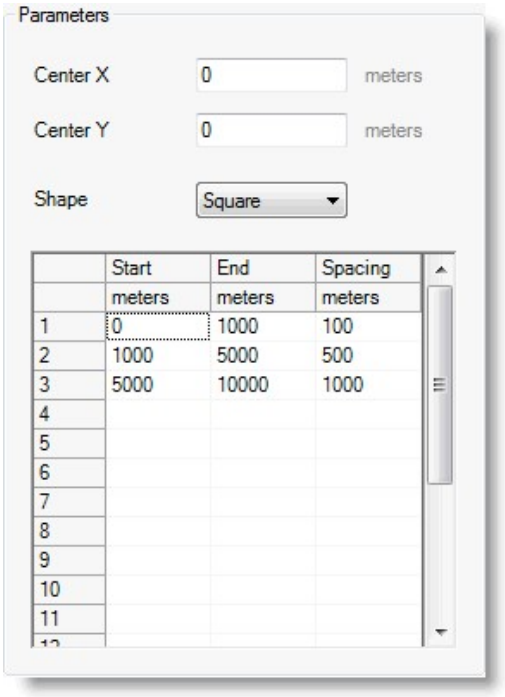


Variable Density

A Variable Density configuration results in an array of model objects distributed throughout a user defined polygon.

An example of a Variable Density model object array is provided here.

Parameters - Variable Density



Center X: The center X coordinate of the grid.

Center Y: The center Y coordinate of the grid.

Shape: Choose either a Square or Circular shape for the grid.

Start: The starting distance from the center of the grid where model objects will begin to appear.

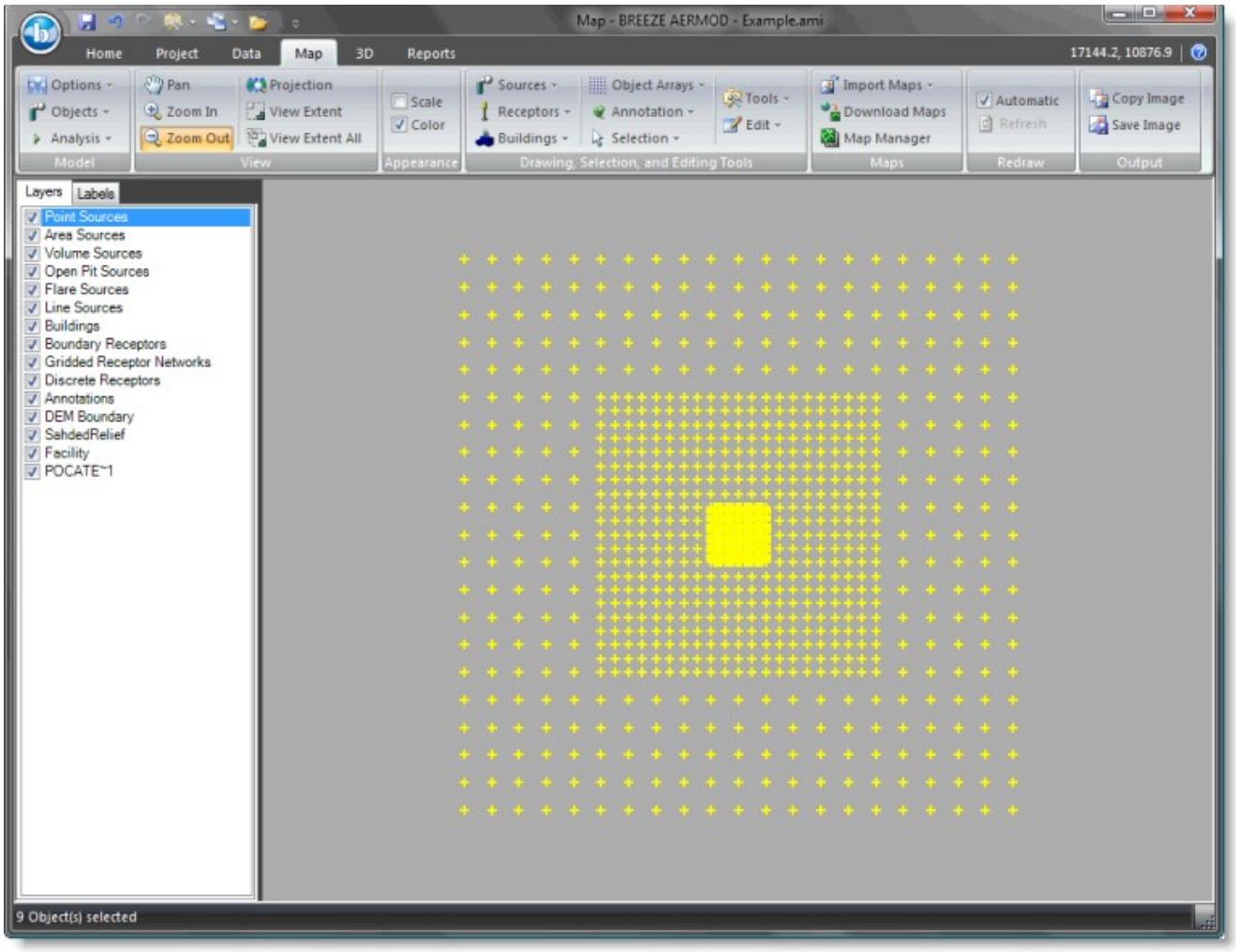
End: The ending distance from the center of the grid where model objects will stop appearing.

Spacing: The spacing of model objects between the Start and End distances for the selected row.

Note: If the Start distance is set to a value that is the same as another Start or End distance, BREEZE will default to the density of the inner grid

Example

A Variable Density configuration results in an array of model objects distributed throughout a user defined area.

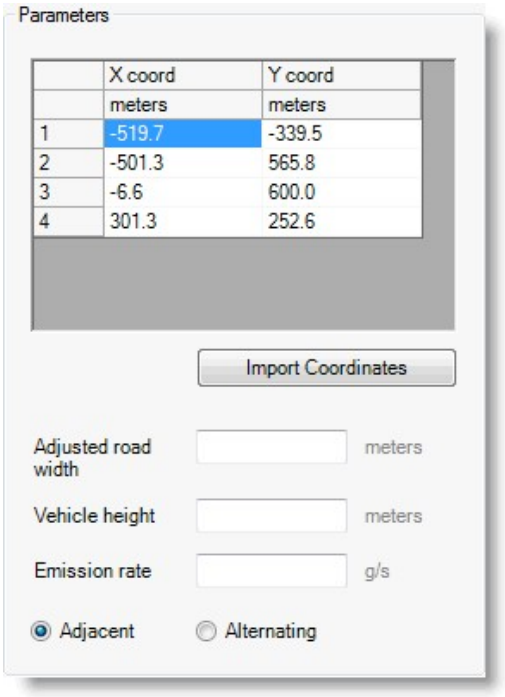


Roadway

A Roadway configuration results in an array of volume sources distributed along a user defined polyline. This is the only object array where the user cannot specify the model object.

An example of a Roadway model object array is provided here.

Parameters - Roadway



X coord: The X coordinate of a corner of the polyline that will represent the road.

Y coord: The Y coordinate of a corner of the polyline that will represent the road.

Adjusted road width: The width of the road plus 6 meters. This is based on current regulatory guidance.

Vehicle height: Height of vehicles using the roadway.

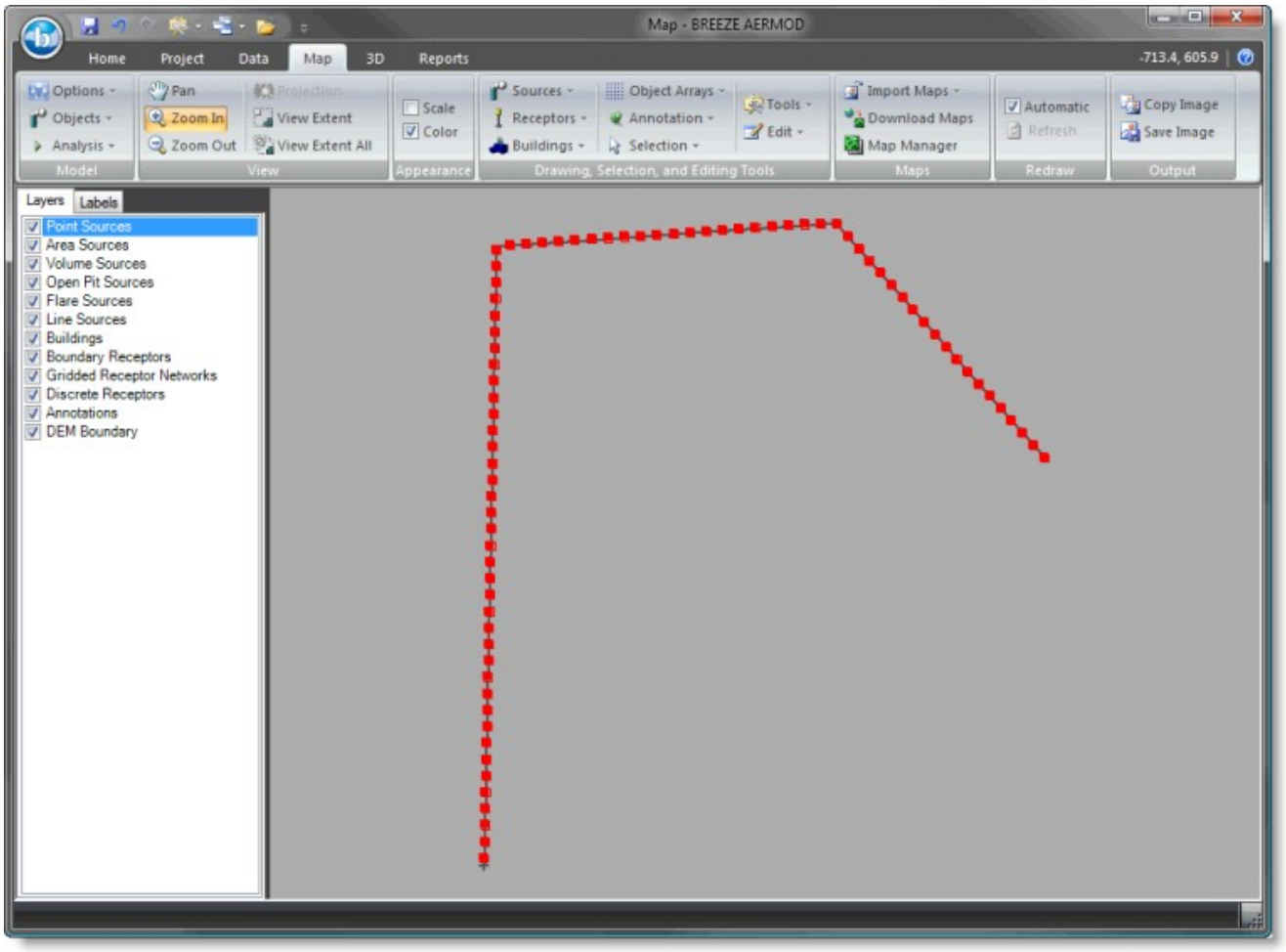
Emission Rate: Emission rate in grams per second.

Adjacent/Alternating: Select how the volume sources should be spaced - adjacent (next to each other) or alternating (with spaces between the sources).

Example

The following is an example of a Roadway model object array using alternating spacing.

This object array is distinguishable from line and polyline volume source object arrays by the black line representing the road.


Map - BREEZE AERMOD
Home
Project
Data
Map
3D
Reports
-713.4, 605.9
Options
Objects
Analysis
Model
Pan
Zoom In
Zoom Out
Projection
View Extent
View Extent All
View
Scale
Color
Appearance
Sources
Receptors
Buildings
Object Arrays
Annotation
Selection
Tools
Edit
Drawing, Selection, and Editing Tools
Import Maps
Download Maps
Map Manager
Maps
Automatic
Refresh
Redraw
Copy Image
Save Image
Output
Layers
Labels
Point Sources
Area Sources
Volume Sources
Open Pit Sources
Flare Sources
Line Sources
Buildings
Boundary Receptors
Gridded Receptor Networks
Discrete Receptors
Annotations
DEM Boundary